Malz++Kassner
GMBH : CAD : SOFTWARE : CONSULTING

CAD6
Release 2015

User's Manual

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Simply good drawing.

Malz++Kassner GmbH • Leopoldstraße 7a • 38100 Braunschweig • Germany • Geschäftsführer: Stefan Malz, Olaf Kassner • Amtsgericht Braunschweig HRB 4986
Fon +49 (0)531 400 137 • Fax +49 (0)531 400 138 • www.malz-kassner.com • post@malz-kassner.com • Deutsche Bank PGK AG, Konto 3611449, BLZ 270 700 24

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Publisher: Malz++Kassner GmbH
Leopoldstrasse 7a
38100 Braunschweig
Germany

Phone +49 531 400 137
Fax +49 531 400 138

www.malz-kassner.com
post@malz–kassner.com

Authors: Timo Bierbaum
Olaf Kassner
Stefan Malz

Design: Stefan Malz

Order Code: 20900.2

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Chapter 1 - Information

Conventions in This Reference

The following conventions are used throughout the reference:

	Very important: You should really read this!
	Note
	Tips & Tricks
Click	The word *click* by itself means click the left mouse button once (sometimes also called left-click). Otherwise we say right-click, or double-click.
SHIFT	This means the Shift key.
KEY1+KEY2	This means press KEY1 and KEY2 at the same time. For example, ALT+TAB means hold down the ALT key while you press the TAB key.
`x+100+2/3`	User input, e.g. text in edit controls.
Italic	Strengthening of single words or phrases.
`MKCAD6.MKD`	File name.
“Options”	When referring to elements in dialog boxes the element names are displayed in quotation marks (e.g. “Options” button).
Shape > Modify Objects	This means the command “Modify Objects” in the “Shape” menu of Malz++Kassner CAD6.

System Requirements

General requirements:

- Windows 8 / 7 / Vista / XP / Server 2012 / Server 2008 / Server 2003
- Mouse or similar input device
- Processor with at least 500 MHz, 512 MB RAM
- XGA graphics with 24 bits color depth, 32 MB video RAM
- 20-70 MB hard disk space

Recommendations for Windows 8 / 7 / Vista / Server 2012 / Server 2008:

- Dual-Core Processor with at least 1 GHz, 1 GB RAM
- SXGA graphics with 32 bits color depth, 128 MB video RAM

Advice for Plotter Output

This section contains important information for Malz++Kassner CAD6 users who wish to output their drawings on an HP-GL plotter. Most standard plotter drivers for Windows have weaknesses. We strongly recommend that you use the Malz++Kassner HP-GL/2 export filter instead. You can use this filter to export your drawing data to an HP-GL/2 file or you can send your drawing data to the output device directly. The filter's various output options allow an exact configuration of the output data. You can download the filter from our Web site (www.cad6.com) for testing.

Some capabilities of the application (filled surfaces, erasers, bitmaps, and clippings) cannot be output to HP-GL plotters, or are limited on HP-GL/2 plotters!

Using Digitizers

Working with Malz++Kassner CAD6 can be made a lot easier by using a digitizer (also known as a “graphics tablet” or simply a “tablet”). A digitizer allows faster and more accurate input than a conventional mouse and the digitizer’s large input area can also be used to select commands.

A tablet with a size of 12 ×12 inches (305 × 305 mm) is perfect, however for certain purposes smaller or larger tablets may be better suited. As input device you can basically choose between the mouse-like cursor or a pen. The cursor normally has at least 4 or 5 buttons but some have significantly more. A pen has usually only 3 buttons and one of these buttons is the pen’s pressure-sensitive tip.

General

All major digitizer manufacturers now offer drivers for Windows. A digitizer can be always used as a replacement of the standard mouse (“Relative Mode”), but a digitizer only becomes really productive when used in the so-called “Pen Mode” or “Absolute Mode” (“WYPIWYG” Mode, “where you point is where you go”). If this mode is active a certain tablet area is mapped on the screen, so that any position of the input device inside this tablet area is mapped absolutely to the mouse pointer’s position on the screen. For example, if you position your input device in the lower left corner of this tablet area then also the mouse pointer is displayed in the lower left corner of the screen.

To be able to use a tablet in “Absolute Mode” it has to support the so-called WINTAB® interface. All major digitizer manufacturers (for example Wacom®, Genius®, CalComp®, and Summagraphics®) deliver along with the tablet appropriate drivers. If you have no driver for your tablet or are looking for a newer version you can download the current driver from the manufacturer’s Web site.

Setting up the Tablet

After the successful installation of the tablet driver you have to specify some settings in the tablet driver’s control programs. Usually you can find the corresponding shortcut in the Windows Control Panel. Although the relevant settings have different names in the various manufacturer-specific control programs, you always have to specify the settings listed below. You should not change the default values of the other settings.

After you have specified all necessary settings in the control program you should exit and restart Windows to activate them (maybe this is not always necessary but it also does no harm). Then start CAD6 and choose the commands **Construction > Digitizer > Button and Tracing Options** and **Construction > Digitizer > Load Assignment** to fit CAD6 to the tablet. The CAD6 “Digitizer” menu is only available if the tablet driver is installed properly.

Position Mode (Control Program only)

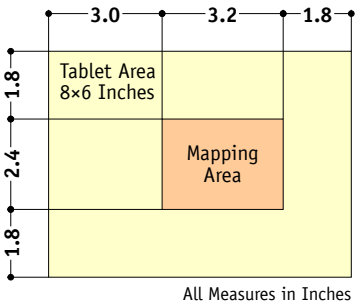
Activate “Pen Mode” or “Absolute Mode” (“WYPIWYG” Mode, “where you point is where you go”) respectively.

Tablet Area (Control Program and CAD6)

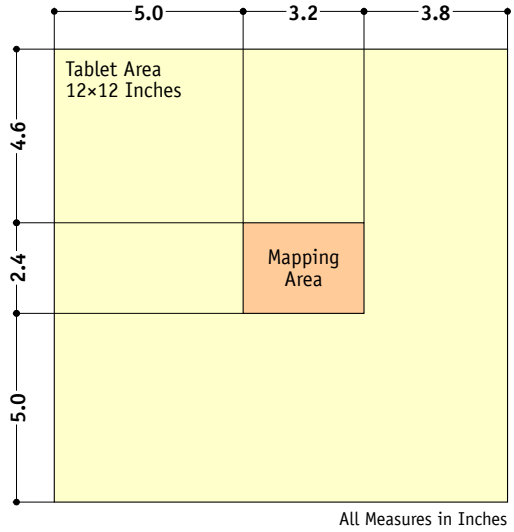
Set the option “Portion of Tablet” or “Enable Mapping” respectively and enter the measures of the mapping area shown in the graphic corresponding to your tablet’s size. For that it is important that you find out whether the measures of the mapping area in your control program have to be entered relative to the lower left or upper left corner of the tablet. You also have to enter the measures of the mapping area in the dialog of the command **Construction > Digitizer > Button and Tracing Options**.

In the tablet driver’s control program you must enter all measures in inches (sometimes you have to specify inches as current unit first) because the tablet sizes are also given in inches. In the dialog of the command **Construction > Digitizer > Button and Tracing Options** all measures are specified in 1/1000 inches relative to the lower left corner. So for a 12 ×12 inches tablet you have to enter `5000` for the distance of the bottom margin.

Measures for 8 × 6 inches tablets:



Measures for 12 × 12 inches (or larger) tablets:



Screen Area (Control Program only)

Activate the setting “Whole Screen” or “Complete” respectively. Sometimes you also have to enter the resolution of you screen, e.g. 1280 × 1024 pixels.

Define Buttons of the Input Device (Control Program and CAD6)

5-Button Crosshair Cursor (arranged in two rows, e.g. Wacom):

	Driver Settings	*CAD6 Settings*
Button 0	Left Mouse Click	Left Mouse Button + Command Level 1
Button 1	Middle Mouse Click	Middle Mouse Button
Button 2	Right Mouse Click	Right Mouse Button + Command Level 2
Button 3	None or Application Defined	Coordinate Tracing
Button 4	None or Application Defined	Coordinate Tracing

4-Button Crosshair Cursor (arranged in one row, e.g. CalComp):

	Driver Settings	*CAD6 Settings*
Button 0	Left Mouse Click	Left Mouse Button + Command Level 1
Button 1	Nothing or Application Defined	Coordinate Tracing
Button 2	Middle Mouse Click	Middle Mouse Button
Button 3	Right Mouse Click	Right Mouse Button + Command Level 2

3-Button Pen:

	Driver Settings	*CAD6 Settings*
Button 0 (tip)	Left Mouse Click	Left Mouse Button + Command Level 1
Button 1	Double Left Mouse Click	Nothing
Button 2	Right Mouse Click	Right Mouse Button + Command Level 2

Load Assignment and Print Template (only CAD6)

Choose the **Construction > Digitizer > Load Assignment** command and load one of the digitizer assignment files located in the `Documents\Malz++Kassner\CAD6\Setting` directory. Which file to load depends on your digitizer's size:

8 x 6 inches digitizer: `DIGIT06.DIG`
12 x 12 inches digitizer (or larger): `DIGIT12.DIG`

Finally, you should load and print the command template for your digitizer. This template contains all predefined command fields available. Malz++Kassner CAD6 normally uses only a small area in the center of the digitizer to map the screen. The remaining area is divided into small square fields, each of which is linked to a command. If a field is clicked on, the associated command is carried out as if it had been selected from the menu.

The digitizer templates are normal drawings that can be printed using the **Print Drawing** or the **Print Section** command. Depending on the digitizer's size, load the following drawing file from the `Documents\Malz++Kassner\CAD6\Setting` directory:

8 x 6 inches digitizer: `DIGIT06E.MKD`
12 x 12 inches digitizer (or larger): `DIGIT12E.MKD`

Print out the drawing with the "Print to multiple sheets" option enabled, trim the sheets, and fasten them together with sticky tape. Place the digitizer menu under the plastic cover in the lower left corner of your digitizer.

The template also contains a graphic and a brief description on how to configure a digitizer.

Deleting or Changing the Command Assignment

If you wish to change or delete the default command assignment, you can do so by using the **Define Digitizer Command Field** or **Delete Digitizer Command Field** commands. These commands are not shown in the menu but are selected by clicking on a command field on the digitizer while at the same time keeping the SHIFT key or the SHIFT+CTRL keys pressed.

Tracing Templates

A common use for digitizers is tracing templates. If a cursor (sometimes with an attached magnifier) is used, it is possible to trace with an precision of up to 1/1000 inches (0.0254 mm).

Malz++Kassner CAD6 can be used for tracing by reserving a button on the input device for it. Using the **Tracing Calibration** and **Set Tracing Origin** commands the digitizer can be set up accurately for this.

Afterwards points on the template can be transferred accurately and easily to a drawing simply by clicking on them with the corresponding button of the digitizer's input device.

Chapter 2 - Introduction

Data Display

Malz++Kassner CAD6 is a vector oriented technical drawing and illustration system. The term “vector oriented” refers to a particular method of displaying data and objects which, because of its high precision and independence from the screen and output resolution is always used for CAD systems and illustration programs.

A line in Malz++Kassner CAD6 is described by the coordinates of its starting and ending points. A 2D coordinate consists of X and Y values:

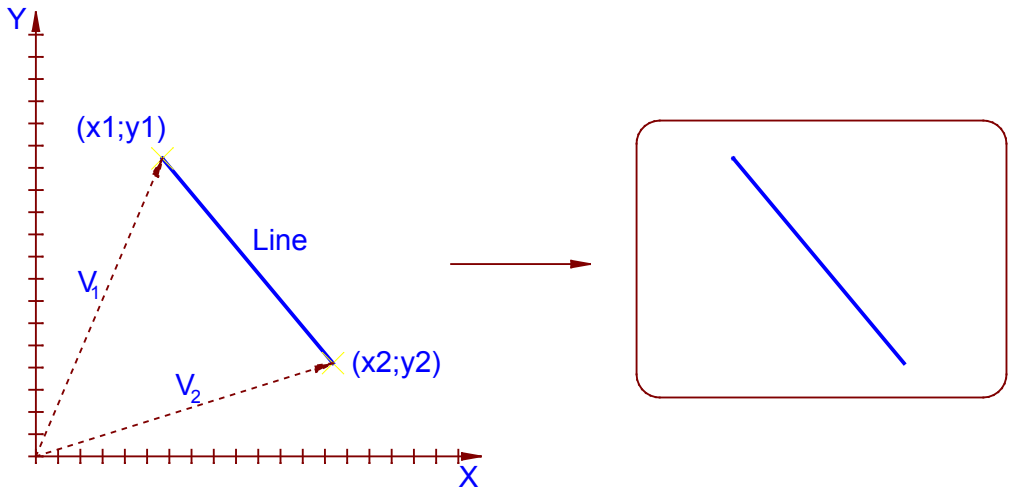


Figure 1: Mathematical display of a line

A vector is uniquely defined by its length and direction. These details can be used to give each point within a layer a unique description.

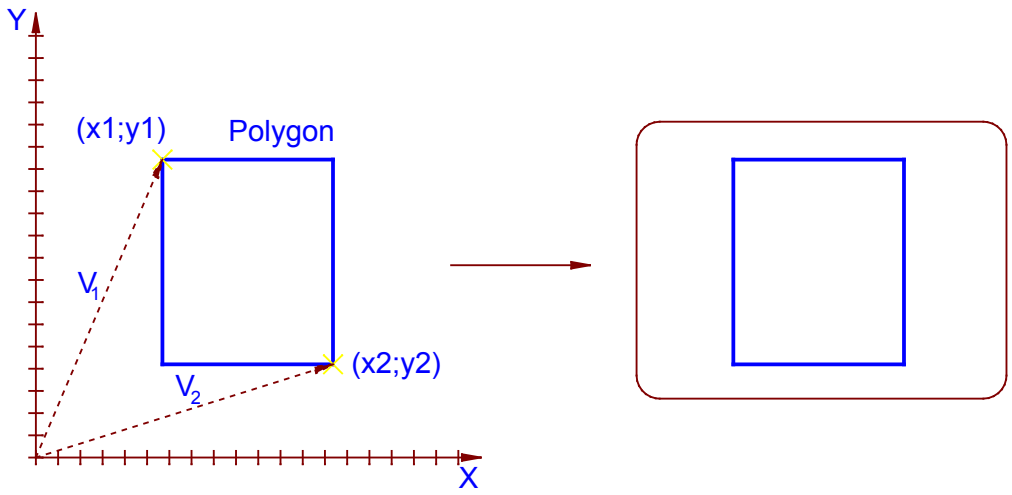
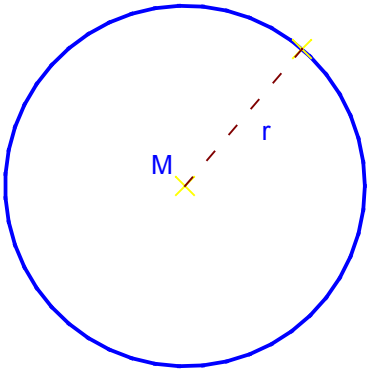


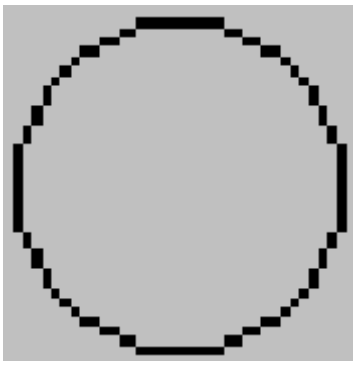
Figure 2: Mathematical display of a rectangle

A rectangle is described by the coordinates of two diametrically opposite corner points (figure 2). This shows one of the important characteristics of vector graphics: the number and complexity of the objects in a drawing and not their size is responsible for the file size.

For output on your screen or printer, the object data is recalculated by the driver according to the relevant resolution.



Data in memory



Data on screen/printer

Figure 3: Recalculation to the output resolution

This procedure may seem unnecessarily complicated, but it has the great advantage of only carrying out the data conversion immediately before output. The points can be calculated with much greater precision within the computer than they can be displayed or printed with. This ensures that each graphic can be handled with the maximum precision (depending on the resolution of the output device).

A further advantage of this method is that objects can be modified without problems. In a vector graphic for example it is easy to scale a circle by changing the radius. In a bitmap graphic on the other hand, the “old” circle has to be deleted and a new one has to be drawn. Or you have to scale the area that contains the circle. But bitmap scaling always yields poor results, e.g. jagged outlines etc.

Coordinate Systems

The basis for screen display is a method of describing the position of each point in your drawing. To do this, a coordinate system has to be defined. With the help of a coordinate system, each point in a drawing can be given a unique description. The best known type of coordinate system is the *cartesian coordinate system*. This consists of a coordinate origin and two coordinate axes at right angles to one another which meet at the origin. Usually, these axes are drawn horizontally and vertically. The horizontal axis is known as the X-axis and the vertical axis as the Y-axis (figure 1).

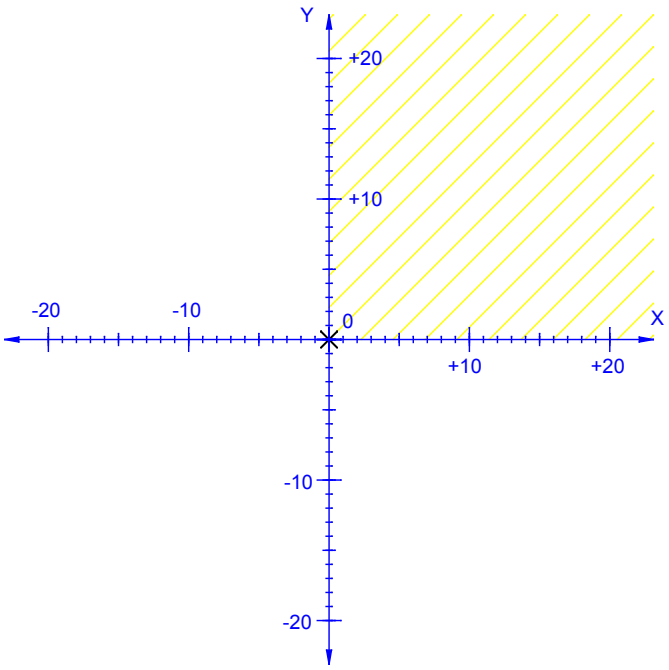


Figure 1: Cartesian coordinate system

Malz++Kassner CAD6 offers distorted forms of coordinate system to aid drawing in isometric and dimetric perspectives. These coordinate systems differ mainly by having a predetermined rotation angle and an altered height/width ratio. Further information on these coordinate systems can be obtained from the reference and the example on dimetric drawing.

Determination of Coordinates

Further observations confine themselves to cartesian coordinate systems. Figure 1 shows such a coordinate system. The X-axis is numbered from left to right, and the Y-axis from bottom to top. In the standard Malz++Kassner CAD6 drawing window, the origin (that is, the point at which X and Y are 0) at the bottom left of the page. The area available to you is the area which is shaded on the graph. The origin can be moved to another point on the page which will make it possible to draw in the other areas.

Various ways can be used to identify a point uniquely. The simplest way is the use of *absolute coordinates*. Absolute coordinates specify the precise X and Y values of a point (Figure 2).

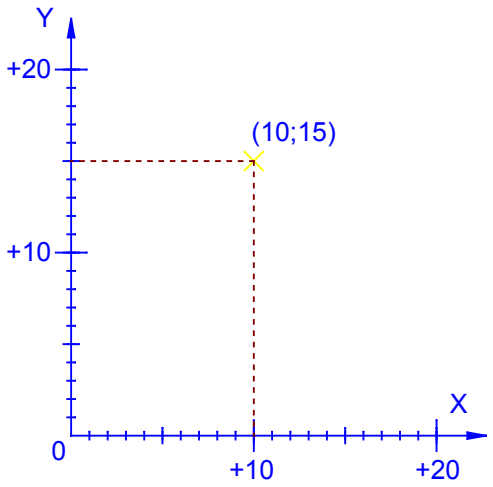


Figure 2: Absolute coordinates

It is possible to identify a point by specifying its position relative to another point. These are called *relative coordinates*. The first point is treated as if it were the origin and the distance between the X and Y values of each point is given. In mathematics, this is often abbreviated as dx and dy. When (x1;y1) are the absolute coordinates of the first point and (x2;y2) are the absolute coordinates of the second point, the position of the second point relative to the first can also be described as follows:

(x2;y2) = (x1+dx;y1+dy)

(Figure 3).

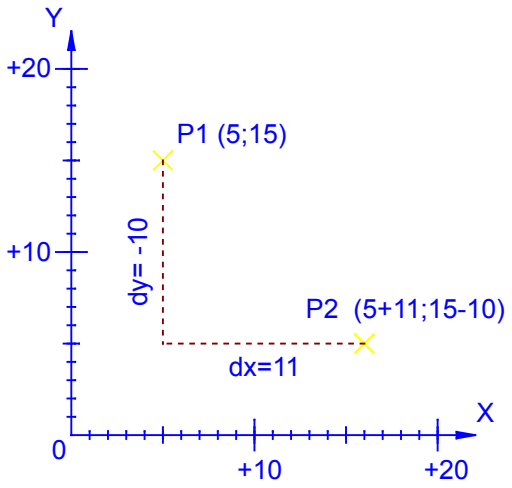


Figure 3: Relative coordinates

A third method is the use of *polar coordinates*. These coordinates are described by distance from the origin and angle from the zero-angle. The mathematical zero-angle (that is, 0°) is along the positive X-Axis, that is, at the “three o’ clock” position (Figure 4).

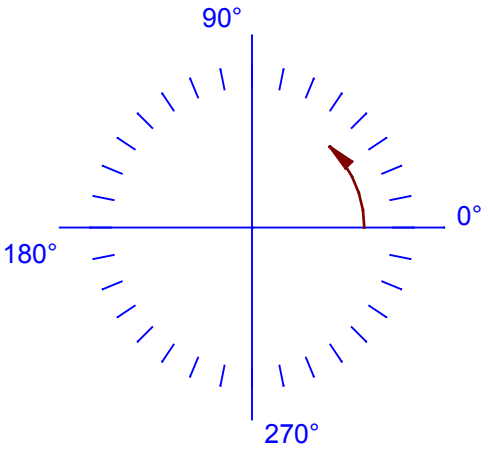


Figure 4: Mathematical zero-angle

Angles are measured in an anti-clockwise direction. This form also describes the position of each point uniquely (Figure 5).

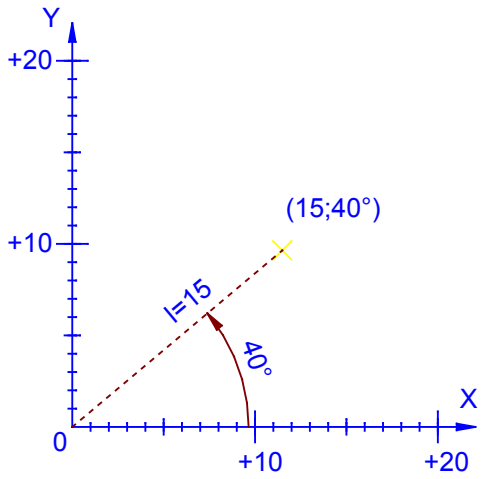


Figure 5: Polar coordinates

Layers, Pens, and Transmission

Each object, e.g. a circle, has its own property set consisting of line width, line pattern, line color, fill color, and fill mode. These properties determine the appearance of the object on the screen and later on the paper.

In addition, each object contains a reference to a layer and a pen, i.e. it lies in a certain layer and uses a certain pen. When the program determines how to draw an object, this not only depends on the object's own properties, but also on the properties transmitted by the assigned layer and the properties specified for the assigned pen.

Both the property set of the layer and the one of the pen do not permanently alter the properties of the object itself, but do only overwrite them temporarily. As soon as the layer or pen reference is changed, the object's appearance will change accordingly - without necessarily changing its properties.

Usually, layers do not transmit any properties, so they do not influence an object's appearance. In this case, the object's appearance is only determined by the pen it uses, i.e. by the pen's properties (see **View > Pens > Edit**). Those pen properties may be different for screen display and printer output.

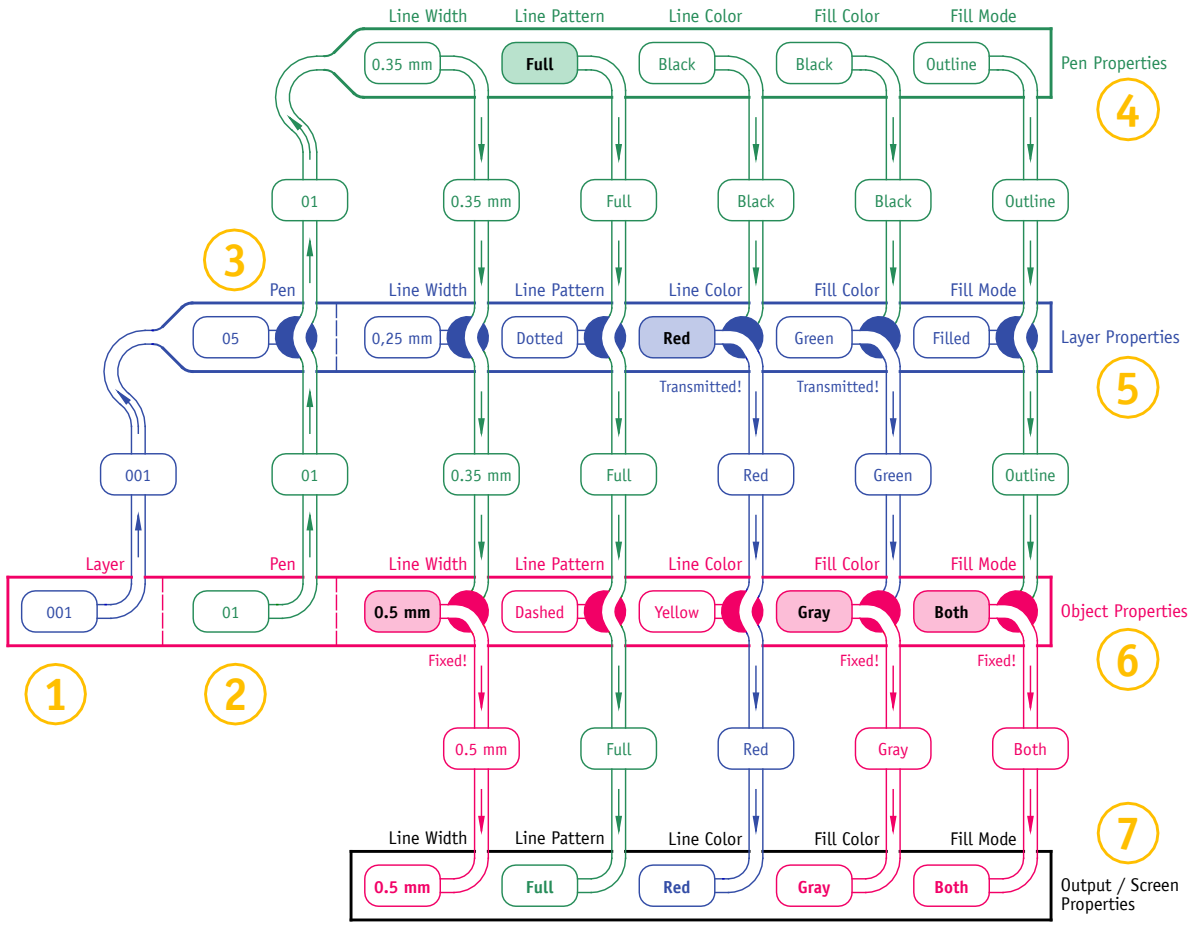
If a layer property is transmitted (see **View > Layers > Edit**), it overwrites the corresponding pen property. It is therefore decisive for the appearance of the object. Layer properties may also be different for screen display and printer output.

Both types of property transmission (from a layer or a pen) can be refused by the object by fixing one or several of its properties. These fixed properties cannot be overwritten by any transmission and will always remain unchanged. Fixing a property will always effect both screen display and printer output.

Layer "*Standard" and pen "*Standard" never transmit any properties. If an object is assigned to the "*Standard" layer and the "*Standard" pen it isn't influenced by changes applied to layers or pens, i.e. the object is completely independent from layer and pen settings. To draw such an object specify "*Standard" as current layer as well as current pen. After drawing the object its properties are then initialized with those currently defined for pen "*Standard".

If you change the properties of pen "*Standard" this hasn't any effect on already existing objects but all objects you draw from then on will be initialized with these new properties. To comfortably edit the properties of pen "*Standard" please select the command **Window > Display Command Bar** and activate there the button groups Line Type, Line Color, Filling Mode, and Filling Color.

To change the properties of a single object or multiple objects use the command **Shape > Edit Properties**.



This graphic illustrates the process used to determine the properties of an object. Each step is shown in the graphic by the relevant number in a circle:

1) First of all, the program determines which layer the object belongs to.
2) Next, the program gets the information about which pen is used by the object.
3) Then the program checks whether or not layer 1 transmits a pen. In such a case the transmitted pen would become the current pen replacing the pen determined in step 2.
4) All pen properties are initially taken from the pen determined in step 2.
5) All properties transmitted by the layer override the corresponding pen properties, as here the line color and the fill color.
6) The program checks each of the object's own properties to see if it is fixed. Fixed object properties are always passed on directly to the output / screen properties. In the diagram line width, fill color, and fill mode are replaced by fixed object properties.
7) Finally the properties determined in step 1 to 6 are used for output to plotter / display on screen.

When drawing an object, it is always assigned to the current layer and the current pen. You can change the current layer by clicking the corresponding button in the **Panel** or by selecting the command **View > Layers > List**. You can also select a new current pen from the **Panel** or by choosing the command **View > Pens > List**.

For some object types, e.g. texts or dimensions, you can optionally specify a default layer (**View > Layers > Defaults**) and a default pen (**View > Pens > Defaults**) which then will be used instead of the current layer or pen respectively when creating such an object.

If you want to assign an object to a different layer or pen later, you can do so by means of the command **Shape > Edit Properties**.

Layers and pens are not only used to change the appearance of objects by property transmission but also to structure a drawing logically. The **Edit Layers** / **Edit Pens** dialog allows you to show or hide layers / pens, to enable or disable them for snapping or modification, etc. If you create layers / pens with meaningful names and consequently assign all newly created objects to these layers / pens, you will find that this feature will greatly simplify your work.

In addition to the properties described above, each object can be assigned to a page (i.e. to one of the pages defined in the drawing, see **View > Pages > Edit**). Once an object is assigned to a page, it will only be output if this page is being output (or displayed on the screen if this page is the active page).

The current pen and layer settings are part of the drawing and are saved with it. To make the altered settings available when the program is restarted save them using the command **View > Template > Save as Default**. In addition to all pens and layers the current page format, pen and layer defaults, the coordinate systems, and the line patterns will be stored as well.

Coordinate Systems, Scales, Grids, etc.

Coordinate Systems

The coordinate systems determine the parameters which the program uses to process drawing data. The coordinate system determines whether you wish to work in a cartesian, isometric or dimetric system. By entering an angle of rotation the orientation of the coordinate system can be displayed. In addition, you can specify measurement units for lengths and angles as well as the type on number, floating point decimals and fractions. The scale is saved with the coordinate system and the position and display grids can be set here. You can set the coordinate system parameters via a dialog called with the command **View > Coordinate Systems > Edit**.

Each drawing can have several coordinate systems defined within it. This can be useful if you have several drawing windows and wish to use a different coordinate system with each one, perform dimensioning in a different coordinate system to that you are drawing in, or use objects with different scales in the same drawing.

To change the current coordinate system apply the corresponding button in the **Ruler** or select the command **View > Coordinate System > List**.

Scales and Units

The most important components of a coordinate system are the scale and the units. The scale determines the “real” size of an object, i.e. how large a drawn object is supposed to be in the model world. Using a scale of 1:20 means that an object whose length is 10 cm on the paper is supposed to have a length of 200 cm in reality.

Above all, scales are influencing dimensions, i.e. dimension statements within a drawing. Dimension do always show the “real” size of an object based on the scale, and not its size on the paper. This is why each dimension is directly linked to a coordinate system, out if which is determines the scaling information. In addition, it uses the desired length and angle units stored in the coordinate system for the dimension.

If dimensions are to be linked to a specific coordinate system, this can either be done in advance by means of the **Lettering > Dimension Parameters** command, or afterwards by means of the **Shape > Edit Properties** command.

Anyway, the scale and the units do not only influence dimensions placed in the drawing, but all types of measurement, both during user entry (e.g. numerical input of a length) and during screen output (e.g. the coordinate display in the **Status Line**).

If you do not like the default measurement units of a coordinate system during input, you can choose different ones (e.g. cm instead of mm) by typing the abbreviation for that unit after the figure. You can use this procedure in all dialogs. You can combine different units in the same calculation (see **Extra > Coordinate Input (F8)**).

Grids

Another important component of coordinate systems are grids, which are divided into position grid and display grid.

The *position grid* is an invisible, regular series of points which the crosshair moves along. The points are always the same distance in one direction from another. The horizontal and vertical divisions can be different. The crosshair 'snaps' from point to point. This ensures that you can only move the crosshair in multiples of the specified grid interval.

The *display grid* shows the grid as small dots on the screen. It can be set separately from the position grid. You can choose a larger display grid than position grid. This speeds up screen redraws considerably and makes for a better overview as not so many points obstruct your view of the objects. If the zoom level is too small, that is if the display grid is too small and the number of points to be shown too large, the display grid is first automatically enlarged and then turned off. As soon as you revert to a level at which the grid can be displayed, it is turned on again.

Both grids are valid for the currently active coordinate system and can be defined as part of it.

The first point of each grid is at the origin. In order to be able to work with a common starting point for the grid in different windows using different coordinate systems, the origin should be moved so that all the grids can be used effectively. To do this, choose the command **View > Coordinate Systems > Set Origin**.

Page Formats

Before starting a new drawing, it is often necessary to specify the page format for the drawing. To do this, choose **View > Pages > Edit**. This calls a dialog where you can either choose a standard page format or a custom page format. The page orientation is also specified in this dialog.

Snapping, Duplicate, and other Options

Snapping

In technical drawing it is not enough to determine the position of points approximately. Most points have to be specified very precisely, which is not possible by hand. Malz++Kassner CAD6 offers mathematical aids for positioning objects. During drawing it is usual to orient yourself to preexisting construction points, such as edges, corners and intersections. To make use of the points within these objects, there are several snapping modes.

If the snap function is active during point entry, then the position of the point will be calculated automatically if it lies within the snap radius. The snap radius is an area around the center of the crosshair. It can be specified in a dialog called by choosing **Extra > Snap Modes > Snap Radius**.

If several points lie within the snap radius determined by the current snap radius settings, the crosshair will always be placed on the nearest one.

The snap function is permanently turned on or off with the F6 key. In addition, snap mode can be briefly turned on with the SHIFT key. Using buttons in the **Panel**, single snapping modes can be toggled.

Duplicate

Every alteration destroys the original object information. The original object information can only be restored by using the UNDO function. To avoid these problems, Malz++Kassner CAD6 has a duplicate function. It is permanently activated or deactivated by pressing the F7 key. The function can be temporarily turned on or off during a command by pressing and holding down the CTRL key. The current setting is shown in the panel and is reversed by holding down the CTRL key.

The duplicate function is a good substitute for the “Copy” command, because it can be combined with every command. The classic “Copy” command can be called by choosing **Shape > Move Object > Standard** and keeping CTRL pressed while placing the objects (i.e. entering the destination point). The duplicate function works similarly with most commands that alter objects.

Multiline

When creating new objects (e.g. drawing a line or circle), the “multiline function” can be used. It determines whether several parallel or concentric objects are created at once (depending on the multiline parameters) or not.

To activate or deactivate the multiline function please choose the command **Duplicate / Multiline (F7)** or click the corresponding button in the **Panel**. To edit the parameters of the multiline function please choose the command **Multiline Parameters** or right-click the corresponding button in the Panel.

The multiline function can be temporarily turned on or off by pressing and holding down the CTRL key.

General Options

The general program options and settings can be edited by selecting the command **Edit > Options > Windows** or **View > Drawing Settings > Screen** respectively.

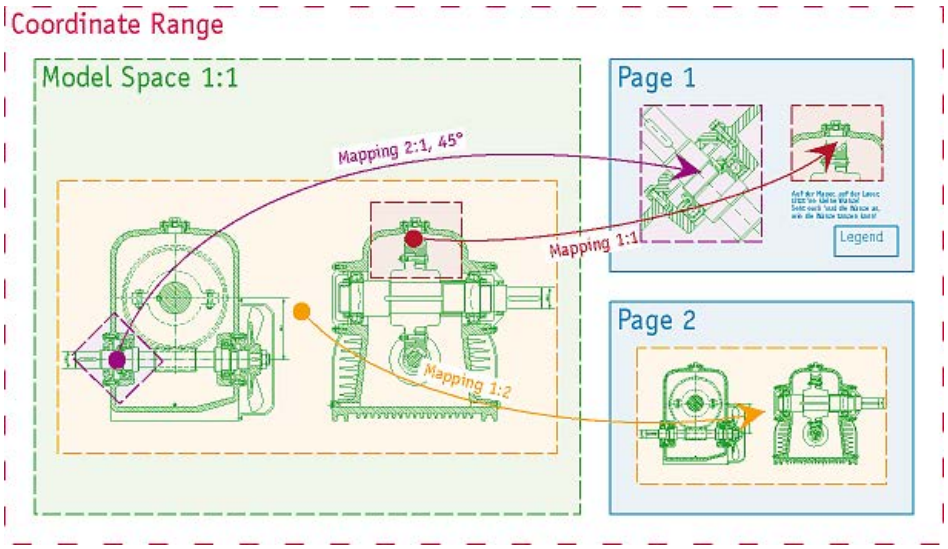
Model Space and Pages

Before starting a new drawing, it is often necessary to specify the page format for the drawing. To do this, choose **View > Pages > Edit**. This calls a dialog where you can either choose a standard page format or a custom page format by double-clicking onto one of the defined pages. The page orientation is also specified in this dialog.

CAD6 supports a Model Space and up to 100 pages in each drawing. Using those, drawings can be structured in different ways depending on complexity and personal liking.

Model Space and Pages Independent of Each Other

If a drawing depicts a „Model“ (e.g. a component, a machine, a ground plan, etc.) of which multiple sections shall later be printed at different places on the paper, maybe even using different scales, it is advisable to place the model completely into the Model Space in order to map parts of it onto the pages later. Here is a schematic view:

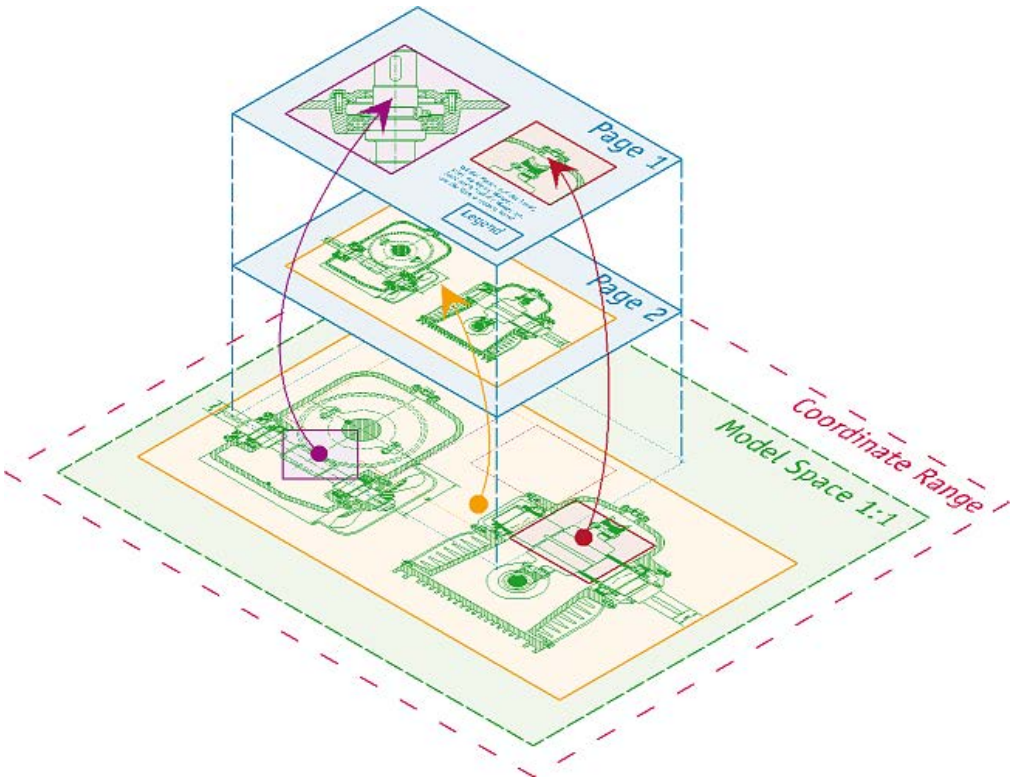


All objects of the model are assigned to Model Space (green). Independent from the final output, the model is usually designed in original size (1:1 scale). Once the drawing of the model has been sufficiently finished, parts of the model will be mapped into one or multiple pages (blue) using Mapping objects. In the respective pages, objects will be added that are either scale-dependent (such as dimensionings) or part of the page's layout (such as legends and drawing frames).

Mappings dynamically project a section of the Model Space within a page (like the image of a supervision camera). Once the Model Space's content changes, the mappings will change, too (at the latest during the next screen redraw). Mappings must always be assigned to pages (not to the Model Space), the objects to be mapped must be part of the Model Space. The objects visible within a mapping can be chosen for purposes such as snapping points or drawing parallels, but they cannot be modified directly. This is only possible in the Model Space.

Since objects in the Model Space and objects in pages are completely independent from each other, they can very well share the same coordinate range, resulting in logical,

stacked planes. You can switch between these planes by switching the current page. Here is a schematic view:



In order to use and output this structure in a sensible way, the following display and output settings should be adjusted:

- Option **View > Page Display > Layout (Active Page / Model)**.

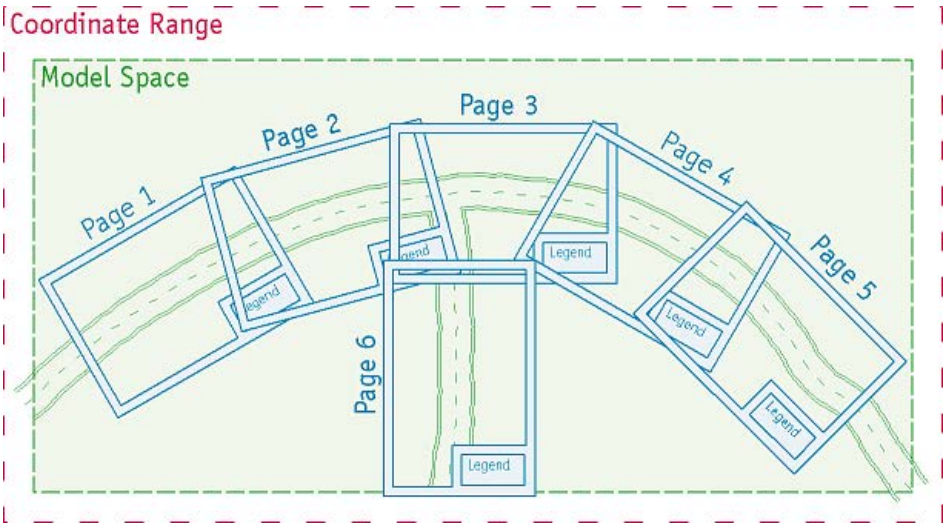
- Command **View > Drawing Settings > Output**,
setting „Output of objects based on their page assignment“:
„Output current page's objects only“.

Furthermore, **View > Pages > Edit** should be called. There, right-click into the list of pages and activate the option „Auto-Assign New Objects to Active Page“ so that it is checked.

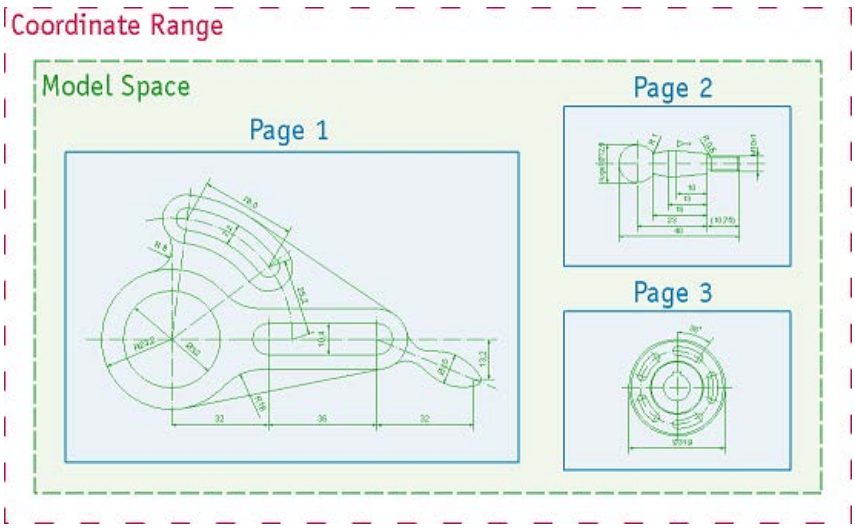
This structure is normally used when importing DXF and DWG drawings.

Pages as Model Space Dividers / Multiple Independent Pages

If the drawing consists of one huge, contiguous model (such as a land register plan, a hall plan, etc.) that shall be printed onto multiple pages, the pages can be used to divide up the Model Space by placing them above the Model Space like punching forms. The pages can be rotated and/or scaled individually. Here is a schematic view:



For multiple, independent models (e.g. a collection of separate components), the same structure can be used, but with non-overlapping pages. Here is a schematic view:



In order to use and output this structure in a sensible way, the following display and output settings should be adjusted:

- Option **View > Page Display > Active Page + Model**.

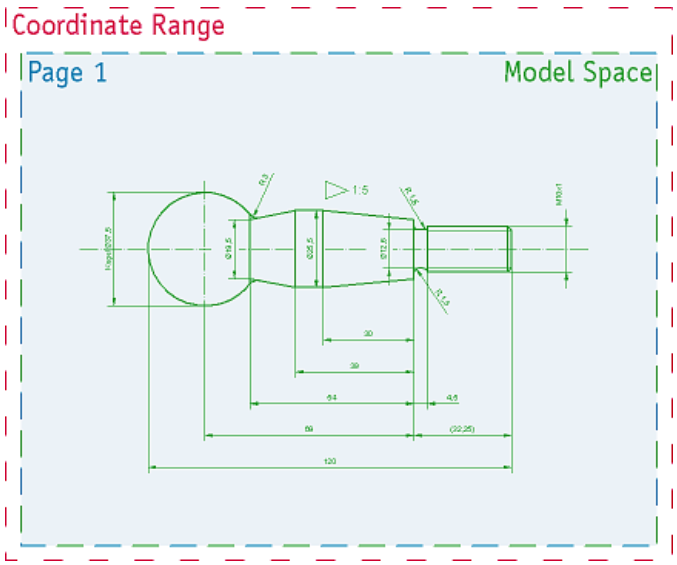
- Command **View > Drawing Settings > Output**,
  setting „Output of objects based on their page assignment“:
  „Output current page + model space“.

Furthermore, **View > Pages > Edit** should be called. There, right-click into the list of pages and clear the option „Auto-Assign New Objects to Active Page“ so that it is unchecked.

In this case, it may be useful to define a separate coordinate system for each page (**View > Coordinate Systems > Edit**) in order to be able to set origin and scale independent for each page. The respective coordinate system can be assigned to its page so that is will automatically be activated when switching between pages.

One Page

Simple drawings that easily fit onto one printer's page can be created completely in model space as long as they lie inside the one page's extents. Here is a schematic view:



In order to use and output this structure in a sensible way, the following display and output settings should be adjusted:

- Option **View > Page Display > Classic (All)**.

- Command **View > Drawing Settings > Output**,
  setting „Output of objects based on their page assignment“:
  „Output all“.

Furthermore, **View > Pages > Edit** should be called. There, right-click into the list of pages and clear the option „Auto-Assign New Objects to Active Page“ so that it is unchecked.

Libraries, Blocks, and Instances

Libraries are organized in a hierarchical structure similar to the file system. A library can contain any number of folders and a folder can hold any number of blocks. Folders are used to group related blocks together.

A block is a collection of objects, like for example lines, rectangles, circles, or texts. Blocks can be used to show screws, furniture, components for local area networks (LANs) etc.

There are a lot of predefined libraries for various areas of application available, e.g. mechanical engineering, electrical engineering, interior design, and others. And of course you can also create your own libraries.

The library / block concept not only allows you to efficiently organize and access frequently used graphical data it also saves a lot of memory. When inserting a block in the drawing you actually don't insert the block's object data but a reference to that block, the so-called instance. I.e. the program inserts only the block name, the library name, and some display parameters (e.g. position, scaling, rotation). Whereas the actual graphical data continues to be stored only in the block definition in the library.

When drawing a block the program uses the block and library name stored in the instance to locate the block definition in the library. The program then uses the graphical data in the block definition and the display parameters stored in the instance to draw the block. Because normally an instance requires much less storage than the actual block data, this concept dramatically reduces memory consumption.

A library is an external file (`*.MKL`) which is independent from the drawing. To use a block (**Library > Block > Insert**) of a specific library in a drawing you first have to load that library into Malz++Kassner CAD6 (**Library > Library Management**). Blocks located in such a library are called *external blocks*. Blocks can also be located in a so-called pseudo-library named "*Internal Blocks". The pseudo-library "*Internal Blocks" is not an external file but it is located in the drawing. Consequently the blocks of the pseudo-library are called *internal blocks*.

When creating your own libraries (**Library > Block > Create (Insertion Point)**) you should consider the advantages and disadvantages of both block types. External libraries can easily be used by several people in different drawings. If for example all people in company use the same libraries located on a central server this helps to standardize drawings and also makes it easy to update them. Because if you replace a library with an updated version (of course the block names must remain unchanged) this update automatically will have an effect on each drawing which contains instances of blocks of that updated library simply by reloading the drawing.

Of course if you want to pass on your drawing files to another company or a client you either also have to pass on the used library files or you have to convert the external blocks into internal blocks using the command **Library > Convert External Blocks**. Alternatively you could also use the command **Library > Block > Resolve Block Instances** to replace each instance with the objects that make up the respective external block. But if the drawing contains a lot of instances this will significantly increase the drawing's size.

Generally you should always use external blocks if you plan to use the blocks not only in the current project or if the blocks must be accessed also by others. On the other hand if you know that a certain block makes only sense in a specific drawing then you should make him an internal block.

There is a special kind of internal blocks the so-called groups. To create a group use the command **Shape > Group > Create Group**. This command automatically creates an internal block stored in the "#G" folder of the pseudo-library "*Internal Blocks" from all currently selected objects.

Blocks can also contain attributes. An attribute can be a text or a number. For example, if you have a furniture library you can use the command **Library > Block > Edit** to add a price attribute to every piece of furniture. After finishing the drawing you then can use the command **Library > Generate Parts List** to generate a list of all pieces and their prices. Use the command **Shape > Edit Text** to edit the attributes of an instance.

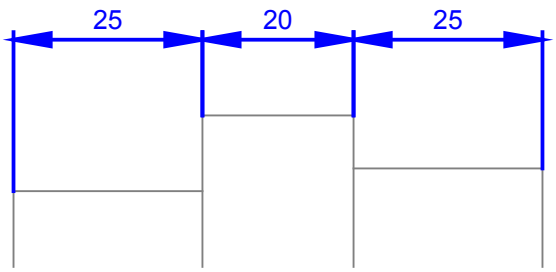
For information on attribute display see **Variables and Attributes**.

Dimensionings

Dimensionings are used in technical drawings in order to display measures of elements that are required to manufacture the drafted object. Even though the measure itself is identical, there are several ways to display such a measure.

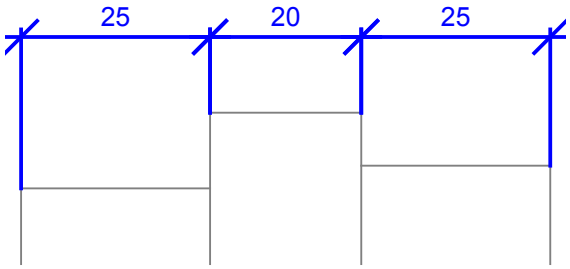
Two standard types of dimension display forms are used most drawings. Once a dimension command has been activated (e.g. **Text > Dimension > Length, Points**), the parameters of this command can be edited either by calling the **Extras > Edit Parameters (+ESC)** command or by pressing SHIFT+ESC.

The first standard method, using arrows as end symbols, is mainly used in engineering. The dimension line is terminated by a filled arrow on both sides, and it is aligned to the extension lines that connect the dimension line with the measured object:



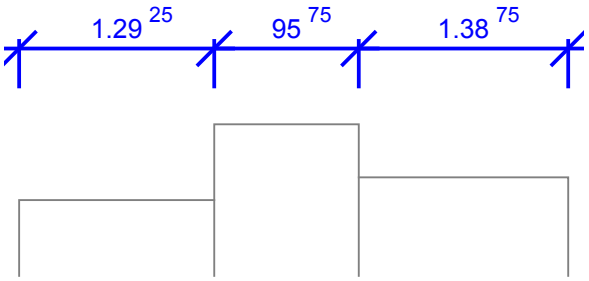
In order to create such a dimension, the dimension line end must be displayed in the “Aligned” mode, the end symbol is a filled arrow.

The second standard display method is used for architectural dimensions. It uses sloping lines or filled circles instead of filled arrows, and the dimension line juts out the extension lines:



Architectural dimensions often use a modified number display mode that automatically converts values below one meter into centimeters and displays values below one centimeter in superscript digits. You can active this number display mode either in the coordinate system (see **Coordinate Systems, Scales, Grids, etc.**) that is assigned to the dimension (button “Numbers”, entry “Length Display”), or directly in the dimension parameters that you can reach as described above.

A typical architectural dimension looks like this:



Each dimension is assigned to a coordinate system from which it retrieves the required information about the scale, length and angle units, and the desired number display mode. The assignment of the coordinate systems is also done in the dimension's parameters. By using multiple coordinate systems you can realize different scales and/or dimension types within one drawing.

Text Input

In all text input fields within dialogs of Malz++Kassner CAD6, you have additional functions that are extensions to the usual possibilities in such fields.

Term Evaluation

If the text input field contains a numeric term (see **Term Evaluation**), this term can be evaluated by means of a single keystroke. In single-line input field, the term evaluation is initiated by means of the “Arrow Up” key, in multi-line input fields by means of the “CTRL + Arrow Up” key combination.

If, at the time of the initiation, a partial text is selected, the term evaluation will only cover this selected text.

The result will be displayed by means of a small tooltip windows that disappears after a short period.

History of Previous Inputs

In all text input field, a list of the least recently entered texts can be called. In single-line input field, this is done by means of the “Arrow Down” key, in multi-line input fields by means of the “CTRL + Arrow Down” key combination.

A list of the least recently entered texts will appear, which also contains some additional commands that will be described below.

List of Variables

The list of least recently entered texts (see above) also contains a list of all variables defined by the user. By simple selecting them from the list, the variable can be inserted to the text at the current cursor position. An additional command calls the dialog used to edit these variables (see **Edit > Edit Variables**).

Search within the Field

Finally, some search commands are available that allow finding texts within the current input field. They can either be selected from the list of least recently entered texts or called by means of key shortcuts:

CTRL + F	Opens a dialog in which the text to be found can be entered. If at the time of opening this dialog a part of the text in the field is selected, this partial text will be copied into the dialog.
F3	Searches for the next occurrence of the text in forward direction, starting with the current cursor position or text selection.
SHIFT + F3	Searches for the next occurrence of the text in backward direction, starting with the current cursor position or text selection.
CTRL + F3	Searches in forward direction for the currently selected text, starting behind the text selection.

SHIFT + CTRL + F3	Searches in backward direction for the currently selected text, starting before the text selection.

The last search text is globally saved, so that it remains available in the next dialogs.

Text Formatting and Unicode

Inside any type of text Malz++Kassner CAD6 allows the use of text formatting and Unicode characters. Both will be used analogically to HTML.

Text Formatting

The following code sequences are used analogically to HTML:

	Bold font on.
	Bold font off.
	Italic font on.
	Italic font off.
<s>	Strikeout on.
</s>	Strikeout off.
<u>	Underlining on.
</u>	Underlining off.
<sub>	Smaller, lowered text on (index). The text will be scaled to 70.71% and will be offset by 20% of the line height.
</sub>	Smaller, lowered text off.
<sup>	Smaller, lowered text on (power). The text will be scaled to 70.71% and will be offset by 40% of the line height.
</sup>	Smaller, lowered text off.

The following code sequences are an extension/modification of HTML:

<frac>XXX<sep>YYY</frac>	A fraction with numerator XXX and denominator YYY. If you wish a separating line, simply underline the numerator.
	This sequence should always be inserted by means of the button “Format> Font“ that is available in all relevant dialog windows. This avoids invalid values.
	Restore original font.
<dx *X*>	Horizontal offset of *X* times type-size (“em quad“), with *X* being a floating point number between -1000 and 1000.
</dx>	Resets the horizontal offset to 0 (will only work in vertical text direction mode).
<dy *X*>	Vertical offset of *X* times type-size (“em quad“), with *X* being a floating point number between -1000 and 1000.
</dy>	Resets the vertical offset to 0 (will not work in vertical text direction mode).
<tab *X*>	Jump to tabulator position at *X* times type-size (“em quad“), with *X* being a floating point number between 0.001 and 1000.
<size *X*>	Relative scaling of the font by factor *X*, with *X* being a floating point number between 0.001 and 1000.
</size>	Restore original scaling. This only reverses explicit scaling set using <size>.
<mode *X*>	Filling mode of the text, with *X* being an integer value stating the filling mode: 0 = framed, 1 = filled, 2 = filled & framed, 3 = erased, 4 = erased & framed.

</mode>	Restore original filling mode.
<fill *R/G/B*>	Filling color of the text in RGB notation, with *R*, *G*, and *B* being unsigned values between 0.0 and 1.0 inclusively, each defining the red, green, or blue component of the color.
<fcmyk *C/M/Y/K*>	Filling color of the text in CMYK notation, with C, M, Y, and K being unsigned values between 0.0 and 1.0 inclusively, each defining the cyan, magenta, yellow, or black component of the color.
<fgray *G*>	Filling color of the text in GRAY notation, with G being unsigned values between 0.0 and 1.0 inclusively, defining the gray component of the color (0=black, 1=white).
<fhsb *H/S/B*>	Filling color of the text in HSB notation, with H being an unsigned value between 0 and 360 inclusively, S and B being unsigned values between 0.0 and 1.0 inclusively, each defining the hue, saturation, or brightness component of the color.
<flab *L/A/B*>	Filling color of the text in LAB notation, with L being an unsigned value between 0 and 100 inclusively, A and B being signed values between -127 and 127 inclusively, each defining the L*, a*, or b* component of the color.
</fill>	Restore original filling color.
<line *R/G/B*>	Outline color of the text in RGB notation, with *R*, *G*, and *B* being unsigned values between 0.0 and 1.0 inclusively, each defining the red, green, or blue component of the color.
<lcmyk *C/M/Y/K*>	Outline color of the text in CMYK notation, with C, M, Y, and K being unsigned values between 0.0 and 1.0 inclusively, each defining the cyan, magenta, yellow, or black component of the color.
<lgray *G*>	Outline color of the text in GRAY notation, with G being unsigned values between 0.0 and 1.0 inclusively, defining the gray component of the color (0=black, 1=white).
<lhsb *H/S/B*>	Outline color of the text in HSB notation, with H being an unsigned value between 0 and 360 inclusively, S and B being unsigned values between 0.0 and 1.0 inclusively, each defining the hue, saturation, or brightness component of the color.
<llab *L/A/B*>	Outline color of the text in LAB notation, with L being an unsigned value between 0 and 100 inclusively, A and B being signed values between -127 and 127 inclusively, each defining the L*, a*, or b* component of the color.
</line>	Restore original outline color.
<afill X>	Opacity (alpha value) of filling with X being an unsigned value between 0.0 and 1.0 inclusively.
</afill>	Restore original filling opacity. This only reverses explicit opacity set using <afill>.
<aline X>	Opacity (alpha value) of outlines with X being an unsigned value between 0.0 and 1.0 inclusively.
</aline>	Restore original outline opacity. This only reverses explicit opacity set using <aline>.
<width *X*>	Line width of X millimeters, with X being a floating point number between 0.001 and 1000.
</width>	Restore original line width.
<<	Output the explicit character '<'.

\~	Output the explicit character '~'.
\{	Output the explicit character '{'.
\}	Output the explicit character '}'.
\\	Output the explicit character '\'.

Examples (base font is "Arial"):

Code sequence in the text:

```
The <b>quick</b> brown fox <u>jumps</u> over the <i><b>lazy</b> dog</i>
```

Displayed text:

The **quick** brown fox <u>jumps</u> over the ***lazy*** *dog*.

Code sequence in the text:

```
(a<sub>1</sub>+a<sub>2</sub>)<sup>2</sup>+
(b<sub>1</sub>+b<sub>2</sub>)<sup>2</sup>=
(c<sub>1</sub>+c<sub>2</sub>)<sup>2</sup>
```

Displayed text:

$(a_1+a_2)^2+(b_1+b_2)^2=(c_1+c_2)^2$

Code sequence in the text:

```
sin(<font 1,512,400,"Symbol">a</font>)<sup>2</sup>+
cos(<font 1,512,400,"Symbol">a</font>)<sup>2</sup>=
sin(<font 1,512,400,"Symbol">b</font>)<sup>2</sup>+
cos(<font 1,512,400,"Symbol">b</font>)<sup>2</sup>=1
```

Displayed text:

$\sin(\alpha)^2+\cos(\alpha)^2=\sin(\beta)^2+\cos(\beta)^2=1$

Code sequence in the text:

```
<frac><u> 1 </u> <sep>16 </frac>a+
<frac><u> 1 </u> <sep>16 </frac>b=
<frac><u> 1 </u> <sep>16 </frac>(a+b)
```

Displayed text:

$\frac{1}{16}a+\frac{1}{16}b=\frac{1}{16}(a+b)$

Unicode Characters

A Unicode character is represented by a code sequence analogically to HTML:

`&#X;` Representation of the Unicode character *X*, with *X* being the Unicode character's number in decimal notation.

Example:

`α` The small alpha (α) of the Greek alphabet, Unicode character 945.

You as a user will not see this representation of Unicode characters since all dialogs in the program which handle text have an automatic translation of Unicode characters.

Variables and Attributes

Malz++Kassner CAD6 supports variables (valid for the total drawing) and attributes (variable only valid within blocks). These variables are in fact text variables because variables can contain numbers, texts, and statements (refer to **Statements in Texts**).

Basics

A variable has to be enclosed in tilde characters ~~ (Ansi 126) to be evaluated in texts or statements respectively, e.g. ~Variable~. The text

```
Price: ~Price~ Euro
```

contains a variable named "Price". If the variable "Price" exists it is evaluated, i.e. the string "~Price~" is replaced by the value or the content of the variable "Price". For example, if the variable has the value 29.95 the following text is created:

```
Price: 29.95 Euro
```

If a text contains a variable that does not exist then this variable cannot be evaluated. In such a case the variable is replaced by the text (***UNDEFINED***) . For the example above the resulting text would be:

```
Price: (***UNDEFINED***) Euro
```

To output the explicit tilde character please use \~.

Customizable Drawing Variables

Drawings can contain any variables that can be used throughout the drawing. These variables can be defined by means of the **Edit > Edit Variables** command.

Example:
Code sequence in the text:
`File Version: ~Version~`

Possibly displayed text:
`File Version: 6.2.0`

Please note that variables have to be enclosed in ~ characters to be evaluated, for example:

```
~Version~
```

Fixed Drawing Variables

Fixed drawing variables always exist and are automatically initialized by CAD6. They cannot be changed by the user.

The following fixed drawing variables are available:

%A0	or	%a0	Name of the author who created the drawing
%A1	or	%a1	Name of the author who last saved the drawing
%A2	or	%a2	Name of the author who is just working with the drawing
%N0	or	%n0	File name of the serial printing database (including path)
%N1	or	%n1	Title of the serial printing database
%N2	or	%n2	Number of data records for serial printing
%N3	or	%n3	Current data record for serial printing
%D0	or	%d0	Creation date of the drawing
%D1	or	%d1	Last saved date of the drawing
%D2	or	%d2	Current date
%D3	or	%d3	Current year (1998, 2006, ..)
%D4	or	%d4	Current month (01..12)
%D5	or	%d5	Current day of month (01..31)
%D6	or	%d6	Current hour (00..23)
%D7	or	%d7	Current minute (00..59)
%D8	or	%d8	Current second (00..59)
%P0	or	%p0	Number of pages
%P1	or	%p1	Total number of sheets in multiple sheets printing mode
%P2	or	%p2	Current sheet number in multiple sheets printing mode
%P3	or	%p3	Page number (this object lies in)
%P4	or	%p4	Width of the page (this object lies in) in [mm]
%P5	or	%p5	Height of the page (this object lies in) in [mm]
%P6	or	%p6	Width of the page (this object lies in) in [inch]
%P7	or	%p7	Height of the page (this object lies in) in [inch]
%P8	or	%p8	Comment of the page (this object lies in)
%S0	or	%s0	Scale of the active coordinate system
%S1	or	%s1	Output scaling factor (as entered in printing dialog)
%S2	or	%s2	Output scaling factor (as defined by page scaling)
%S3	or	%s3	Output rotation angle (as defined by page rotation) in [deg]
%V0	or	%v0	First dimension value (in any type of dimension)
%V1	or	%v1	Second dimension value (only in a coordinate dimension)
%X0	or	%x0	File size in KB (in memory)
%X1	or	%x1	Number of objects in the drawing
%X2	or	%x2	Number of internal blocks in the drawing
%X3	or	%x3	Number of used external blocks
%X4	or	%x4	Number of used bitmaps
%X5	or	%x5	Number of used external references
%Z0	or	%z0	File name of the drawing (including path)
%Z1	or	%z1	Title of the drawing
%Z2	or	%z2	Topic of the drawing
%Z3	or	%z3	Comment on the drawing
%Z4	or	%z4	File name of the drawing (without path)
%Z5	or	%z5	File name of the drawing (without path and extension)

Example:

Code sequence in the text:
```
Page ~%p3~ of ~%p0~
```

Possibly displayed text:
```
Page 1 of 16
```

Please note that variables have to be enclosed in ~ characters to be evaluated, for example:

```
~%a0~
```

Database Requests

Database requests are only available if the user has opened the corresponding database by means of the command **Library > Databases** and serial printing is active. Each cell of the database is considered to be a (virtual) variable which can be evaluated by means of a database request.

A database request has one of the following forms:

`%[DatabaseName]ColumnNumber,RowNumber`
: Gets the content of the cell in column `ColumnNumber` (starting with 1) and row `RowNumber` (starting with 1) of the database titled `DatabaseName`.

`%[DatabaseName]“ColumnName“,RowNumber`
: Gets the content of the cell in the column titled `ColumnName` and row `RowNumber` (starting with 1) of the database titled `DatabaseName`.

`%[DatabaseName]ColumnNumber,“FindText“@FindColumnNumber`
: Gets the content of the cell in column `ColumnNumber` (starting with 1) and the row that contains the string `FindText` in column `FindColumnNumber` (starting with 1).

`%[DatabaseName]ColumnNumber,“FindText“@“FindColumnName“`
: Gets the content of the cell in column `ColumnNumber` (starting with 1) and the row that contains the string `FindText` in the column titled `FindColumnName`.

`%[DatabaseName]“ColumnName“,“FindText“@FindColumnNumber`
: Gets the content of the cell in the column titled `ColumnName` and the row that contains the string `FindText` in column `FindColumnNumber` (starting with 1).

`%[DatabaseName]“ColumnName“,“FindText“@“FindColumnName“`
: Gets the content of the cell in the column titled `ColumnName` and the row that contains the string `FindText` in the column titled `FindColumnName`.

If `ColumnName`, `FindText`, or `FindColumnName` starts with as asterisk ‘*’, the search method will be case-insensitive, partial match. If not, the search method will be case-sensitive, complete match.

In order to access the default internal database of the current database, use an asterisk ‘*’ only as database name. To access another internal database, use an asterisk followed by the internal database's name, e.g. “*Partslist1”.

Example:

Code sequence in the text:
`~%[Products]“Description“,“53790“@“Part No“~`

Possibly displayed text:
`Switch, Shape C, small, black`

Explanation:

Technically, the application uses the database titled “Products“. In this database, it first searches for a column titled “Part No“ (i.e. a column containing exactly the phrase “Part No“ in its top cell). In that column, it searches for a cell containing the text “53790“. Once found, it stores the row number where it found that cell. Then, it searches for a column titled “Description“ (i.e. a column containing exactly the phrase “Description“ in its top cell) and takes the content of the cell in this column and the row previously stored.

In plain English: The application searches for the description of the product with part number 53790.

Please note that database requests have to be enclosed in `~` characters to be evaluated, for example:

```
~%[Products]12,15~
```

Serial Printing Fields

Serial printing fields are only available if the user has opened the corresponding database by means of the command **Library > Databases** and serial printing is active. Each cell of the database is considered to be a (virtual) variable which can be evaluated by means of a serial printing field.

Serial printing fields have one of the following forms:

`%#ColumnNumber`
: Gets the content of the cell in column `ColumnNumber` of the current serial print database’s entry.

`%“ColumnName“`
: Gets the content of the cell in the column titled `ColumnName` of the current serial print database’s entry.

If `ColumnName` starts with as asterisk ‘*’, the search method will be case-insensitive, partial match. If not, the search method will be case-sensitive, complete match.

Example:

Code sequence in the text:
`~%“Name“~`

Possibly displayed text:
`Smith`

Explanation:

The application uses the column titled “Name“ (i.e. a column containing exactly the phrase “Name“ in its top cell) of the database selected for serial printing, and returns the column content of the row currently selected by the user (or the serial printing process).

Please note that serial printing fields have to be enclosed in ~ characters to be evaluated, for example:

```
~%“Name“~
```

Attributes

If you want to display attributes of a block, you have to know this *before* you create that block. Let’s assume you want to create a block containing the symbolic representation of an electric resistor. This symbol should contain a text that displays the resistor’s resistance.

The resistance will be stored in a local attribute of the block to be created, as this local attribute may have different values for each instance of the block. In our example, this attribute shall be called “Resistance“.

Now, let’s create the block. First, draw the resistor itself. A rectangle and maybe two line for the connections will do. Then, create a standard text (**Lettering > Text > Standard**) containing just the following short text:

```
~Resistance~
```

Place this text inside the resistor’s rectangle. Please note that this text will also be rotated and scaled if you rotate or move the block itself later on! So make sure the text’s position will still make sense then.

After you have created all components of the resistor symbol, create a block using **Library > Block > Create (Insertion Point)**. In the dialog appearing, set the “Edit Attributes“ check box. After entering a name for the block and pressing “OK“, another dialog appears. Here, add a local attribute named “Resistance“ to this block. Initialize it with any useful value, e.g. “100 ohm“.

After you have created this block, you are ready to use the resistor symbol. Create multiple instances of this block using the **Library > Block > Insert** command. Be sure that the “Copy Attributes“ check box is set in the block selection dialog!

You will see that all resistor symbol placed display the same text “100 ohm“. Now, choose the **Shape > Edit Text** command and identify one of these resistor symbols. In the appearing dialog, change the value of the “Resistance“ attribute to “10 kohm“. After pressing “OK“, the identified resistor symbol will display “10 kohm“ instead of “100 ohm“.

If you want to edit the attribute's value for *each* block you place, you can simply set the "Edit Attributes" check box in the block selection dialog of the **Library > Block > Insert** command. This will display the "Edit Attributes" dialog each time you place a symbol.

Please note that attributes have to be enclosed in ~ characters to be evaluated, for example:

```
~Resistance~
```

Implicit Attributes

Implicit attributes are a special kind of local attributes. They are owned by *any* block, even though they are not visible to the user, and are usually used for parts lists. They are used to store internal information such as the block name, library name, position, etc.

The following implicit attributes are available:

- *00 The name of the block itself.
- *01 The name of the library the block reside in.
- *04 X-coordinate of the block's insertion point in internal millimeters, i.e. relative to the page's center and non-scale-dependent.
- *05 Y-coordinate of the block's insertion point in internal millimeters, i.e. relative to the page's center and non-scale-dependent.
- *06 Rotation of the block in degrees.
- *07 Horizontal scaling of the block relative to 1.
- *08 Vertical scaling of the block relative to 1.
- *09 Horizontal distortion (shearing) of the block in degrees.

Please note that attributes have to be enclosed in ~ characters to be evaluated, for example:

```
~*05~
```

Object Properties

Object properties are variables allowing to access properties of specific objects by means of their unique ID.

The following object properties are available:

- *p0(UniqueID) The surface area of the object with the given unique ID in square millimeters (if available, otherwise 0).
- *p1(UniqueID) The perimeter / length of the object with the given unique ID in square millimeters (if available, otherwise 0).
- *p2(UniqueID) The dimension number of the object with the given unique ID in square millimeters (if available, otherwise empty string).
- *p3(UniqueID) The text in the object with the given unique ID in square millimeters (if available, otherwise empty string).

*p4(UniqueID)	The X-coordinate of the object with the given unique ID in internal millimeters, i.e. relative to the page's center and non-scale-dependent (if available, otherwise 0).
*p5(UniqueID)	The Y-coordinate of the object with the given unique ID in internal millimeters, i.e. relative to the page's center and non-scale-dependent (if available, otherwise 0).

Please note that object properties have to be enclosed in ~ characters to be evaluated, for example:

```
~*p0(0E96B221D35D6F4796CC05A5000F6045)~
```

Miscellaneous

Further variables are used by the program for internal purposes:

#AutoPosNum	An automatically generated position number.

Please note that variables have to be enclosed in ~ characters to be evaluated, for example:

```
~#AutoPosNum~
```

Term Evaluation

In any numerical entry field, such as the **Coordinate Entry (F8)** or the entry of a rotation angle, you can state a mathematical term instead of an explicit number. In addition terms are also evaluated in texts if the corresponding statement is used (refer to **Statements in Texts**).

Terms consist of operands (numbers, variables, or constants), operators, parentheses, and functions. The usual mathematical notation is used. A term is evaluated from left to right (left-associativity).

Operators

The following list shows the precedence of operators starting with the highest (operators in the same group have equal precedence):

An operand may by preceded with a single sign (unary operator):

+ or -	Sign	Example:	+8	(result: 8),
			-2	(result: -2)

This unary operator always yields 0 (operand not equal 0) or 1 (operand is 0) as result

!	Logical NOT	Example:	0	(result: 1),
			12	(result: 0)
^^	Power	Example:	2^^8	(result: 256)
*	Multiplication	Example:	3*4	(result: 12),
			2^^8*2	(result: 512)
/	Division	Example:	8/2	(result: 4),
			8/2*4	(result: 16)
+	Addition	Example:	8+2	(result: 10),
			8+2*4	(result: 16)
-	Subtraction	Example:	10-2	(result: 8)

Operators in this group always yield 1 (true) or 0 (false) as result:

<	Less than	Examples:	1<2	(result: 1),
			8<5	(result: 0)
<=	Less or equal	Examples:	2<=2	(result: 1),
			8<=5	(result: 0)
>	Greater than	Examples:	1>0	(result: 1),
			5>8	(result: 0)
>=	Greater or equal	Examples:	1>=1	(result: 1),
			5>=8	(result: 0)
==	Equal	Examples:	1==1	(result: 1),
			5==8	(result: 0)
!=	Unequal	Examples:	1!=0	(result: 1),
			5!=5	(result: 0)
&	Bitwise AND	Example:	24&15	(result: 8)
|	Bitwise OR	Example:	16|8	(result: 24)
^	Bitwise XOR	Example:	24^31	(result: 7)

Operators in this group always yield 1 (true) or 0 (false) as result, any operand not equal 0 is interpreted as true, 0 means false:

`&&`	Logical AND	Examples:	`1&&!0`	(result: `1`),
			`0&&1`	(result: `0`)
`||`	Logical OR	Examples:	`5||0`	(result: `1`),
			`0||0`	(result: `0`)

By means of parentheses you can enforce another precedence or associativity of operators:

`()`	Examples:	`(8+2)*4`	(result: `40`),
		`8/(2*4)`	(result: `1`)
		`2^^(8*2)`	(result: `65536`),
		`(0!=1)||(8<5)`	(result: `1`)

Functions

Arguments of functions must be always enclosed in parentheses. If a function requires two arguments then these arguments have to be separated by a semicolon. The following functions are available:

`sqr`	Square	Example:	`sqr(4)`	(result: `16`)
`sqrt`	Square root	Example:	`sqrt(256)`	(result: `16`)
`ln`	Logarithm with base e (natural logarithm)			
		Example:	`e^^(2*ln(10))`	(result: `100`)
`sin`	Sinus	Example:	`sin(90)`	(result: `1`)
`cos`	Cosinus	Example:	`cos(0)`	(result: `1`)
`tan`	Tangens	Example:	`tan(45)`	(result: `1`)
`arcsin`	Arcussinus	Example:	`arcsin(1)`	(result: `90`)
`arccos`	Arcuscosinus	Example:	`arccos(1)`	(result: `0`)
`arctan`	Arcustangens	Example:	`arctan(1)`	(result: `45`)
`ceil`	Smallest whole number larger than or equal to the argument			
		Examples:	`ceil(1.9)`	(result: `2`),
			`ceil(2.4-0.5)`	(result: `2`)
`floor`	Largest whole number smaller than or equal to the argument			
		Examples:	`floor(1.9)`	(result: `1`),
			`floor(1.6+0.5)`	(result: `2`)
`round`	Nearest whole number equal to the argument			
		Examples:	`round(1.1)`	(result: `1`),
			`round(2.4-0.5)`	(result: `2`)
`abs`	Absolute value	Examples:	`abs(-1.1)`	(result: `1.1`),
			`abs(-2)`	(result: `2`)
`min`	Minimum	Examples:	`min(-1;2)`	(result: `-1`),
			`min(1.6;2.4)`	(result: `1.6`)
`max`	Maximum	Examples:	`max(-1;2)`	(result: `2`),
			`max(1.6;2.4)`	(result: `2.4`)
`rand`	Generates a pseudorandom number in the range 0 to argument.			
		Examples:	`rand(1)`	(result: `0.6586`),
			`rand(100)`	(result: `17.3681`)

Variables

In case you have defined variables within the active drawing by means of the **Edit > Edit Variables** command you can use those as operands as well. For that you have to enclose the variable name in ~characters (same as with variables in text, see **Variables and Attributes**):

Example: `(~var1~+~var2~)/2` (result: `mean value of var1 and var2`)

Constants

According to the type of input being carried out, certain constants are available which can be used in the terms. They allow, for example, relative values to be entered during coordinate entry. The following constants are available:

`x` and `y`	Current X- and Y-coordinates of the reference point
`l`	Current length of the reference object or previously entered line section
`r`	Current radius of the reference object
`a` and `w`	Current angle of the reference object or previously entered line section
`b`	Current arc length of the reference object
`_fl`	"**F**rame **L**eft"
`_fr`	"**F**rame **R**ight"
`_ft`	"**F**rame **T**op"
`_fb`	"**F**rame **B**ottom" Rectangular coordinates of the frame which surrounds all currently selected objects.
`_pl`	"**P**age **L**eft"
`_pr`	"**P**age **R**ight"
`_pt`	"**P**age **T**op"
`_pb`	"**P**age **B**ottom" Corner coordinates of the current page.
`e`	Base number for natural logarithms (2.7182...)
`pi`	pi (π) (3.1415...)

Units

All values entered without a specific unit relate to the current length or angle unit of the active window's coordinate system.

To specify a unit for a value, the appropriate abbreviation should be used. These are the possible units and their abbreviations.

µm	Micrometer	1/1000 mm
mm	Millimeter	1 mm
cm	Centimeter	10 mm
dm	Decimeter	100 mm
m	Meter	1000 mm
km	Kilometer	1000000 mm
mil	Mil	1/1000 in
in	Inch	1 in = 25.4 mm
ft	Foot	12 in
yd	Yard	3 ft
ml	Mile	1760 yd
dp	Decipoint	1/720 in
pt	Point	1/72 in
bp	Didot	1/2660 m
cic	Cicero	12/2660 m

Possible angle units and abbreviations

deg or °	Degree	360 per rotation
min or ‘	Minute	1/60 degree
min or “	Second	1/3600 degree
gra	Gon	400 per rotation
rad	Radian	2pi per rotation
rel	Relative	1 per rotation

Units can be mixed within a term. These are examples using `mm` as the current length unit and `deg` as the current angle unit.

100+1in	equals	100 mm + 1 in = 125.4 mm
(100+1)in	equals	101 in = 2565.4 mm
2yd+1ft+9,5in	equals	93.5 in = 2374.9 mm
sin(pi rad)	equals	sin(180°) = 0.0
45°60’3600“	equals	47°

Statements in Texts

If enabled by the user (**Drawing Settings > Screen**) statements in texts are evaluated. This can be used, for example, to display the result of a **Term Evaluation**. For that a statement has to be enclosed in braces `{ }`. Additionally the opening brace `{` can be immediately followed by a control character which defines the type of the statement. The result of the evaluation is a text which is then used to replace the corresponding statement text. If the resulting text is empty the statement text is deleted. The assignment and the comment statement have no result text, here the statement text is always deleted.

Overview of Statement Types

Functional Statements:

`{=Var {Term}}`	Assignment statement (term evaluation)
`{=Var {$Text}}`	Assignment statement (text copying)
`{?Condition {Term1}{Term2}}`	Conditional statement (term evaluation)
`{?Condition {$Text1}{$Text2}}`	Conditional statement (text copying)
`{!Var {Term1}{Term2}}`	Non-existence-statement (term evaluation)
`{!Var {$Text1}{$Text2}}`	Non-existence-statement(text copying)
`{$Text} / {$}`	Text or empty text statement respectively
`{*Comment}`	Comment statement

Numerical Term Statements (yield the result of the term evaluation):

`{Term}`	Maximum number of decimal places
`{%Term}`	Rounded
`{.Term}`	Rounded, decimal point
`{,Term}`	Rounded, decimal comma
`{+Term}`	Rounded, decimal point, always with sign
`{&Term}`	Rounded, decimal comma, always with sign
`{#Term}`	Integer

C-Type Statements (yield the result of the term evaluation):

`{f Term}`	Floating-point value
`{g Term}`	Exponential value
`{d Term}`	Signed integer value
`{x Term}`	Unsigned hexadecimal value
`{b Term}`	Unsigned binary value

`Var` here means variable name without tilde characters. `$Text` denotes a text which is not evaluated as a numerical term, but initially copied only. `Condition` is a text which is interpreted as a numerical expression. If the result of this expression is zero then the condition is considered to be *false* (not fulfilled). Any other result means the condition is *true* (fulfilled). If the first condition is true then the first statement (`{Term1}` or `{Text1}`) is executed, otherwise the second. Similarly for the non-existence-statement applies: If the variable does not exist or if it is empty the first statement (`{Term1}` or `{Text1}`) is executed, otherwise the second.

`Term` denotes a text which is evaluated as a numerical term (control characters can be used to change the presentation of the result). The result of a term evaluation is always a number which is written to the result text according to its type (which is defined by the

control character). An empty `{}` term always yields `0.0`. If a syntax error occurs then the result text contains `(***SYNTAX ERROR***)` afterwards. For rounding the number of decimal places set by the user is used. If the decimal separator is not specified by the statement type the active decimal separator is applied.

If the result text is part of a text object which is displayed on screen, printed or exported, then the total text is evaluated as long as there are no more statements. This also applies to the control texts of the CAM postprocessor. That means, the resulting total text does not contains any braces `{ }` afterwards, except those that have been inserted as explicit characters (`\{ \}`).

The spaces in the statement texts are not required, they have been added only for better readability.

Samples

Assignment Statement

`{=Var {Term}}` or `{=Var {$Text}}`

The assignment statement is marked by the control character '=', it is followed by the variable name and a term or text statement. If the specified variable exists, the result of the term evaluation is assigned to that variable or the text is copied to the variable. If the variable does not exist a syntax error is output.

Example 1:
Statement in the text:
`{=Counter {# ~Counter~ + 15}}`

Result text:
None

Explanation:
The variable "Counter" is incremented by 15. The assignment statement does not create an output.

Example 2:
Statement in the text:
`{=String {$This is a text}}`

Result text:
None

Explanation:
The text "This is a text" is copied to the variable "String". The assignment statement does not create an output.

The statement `{=Counter{0}}` assigns zero to the variable “Counter“. The statement `{=String{$}}` assigns an empty text to the variable “String“, i.e. the variable’s content is cleared.

Conditional Statement

`{?Condition {Term1}{Term2}}` or `{?Condition {Text1}{Text2}}`

The assignment statement is marked by the control character ‘?’, it is followed by the numerical condition (which may include a complete term along with variables) and two term or text statements. If the condition is true, i.e. the numerical expression is non-zero, the first statement is used. On the other hand, if the condition is false, i.e. the numerical expression is zero, the second statement is used.

Please note that in order to compensate the internal calculation inaccuracy values between -1e-10 and 1e-10 are also considered to be “false“.

Example 1:
Statement in the text:
`{? ~Value~!=0 {100/~Value~}{0}}`

Result text (variable “Value“ equals 5):
`20.0`
Result text (variable “Value“ equals 0):
`0.0`

Explanation:
If the condition is true (“Value“ is non-zero) the result of the division 100 divided by variable “Value“ is output. Otherwise “0.0“ is output.

Example 2:
Statement in the text:
`{? ~Value~> 5 {$Big}{$Small}}`

Result text:
`Big` oder `Small`

Explanation:
If the condition is true (“Value“ is greater than 5) the text “Big“ is used. Otherwise the text “Small“ is used.

Please note that the two text statements may include statements again.

Non-Existence Statement

`{!Var {Term1}{Term2}}` or `{!Var {Text1}{Text2}}`

The assignment statement is marked by the control character '!', it is followed by the variable name and two term or text statements. If the variable is not defined or empty the first statement is used. If the variable does exist and if it is not empty the second statement is used.

Example 1:
Statement in the text:
`{!Value {0}{100*~Value~}}`

Result text if variable "Value" does not exist:
`0`
Result text if variable "Value" equals 0.75:
`75,0`

Example 2:
Statement in the text:
`{! %"Name" {$No Name}{$~%"Name"~}}`

Result text if the serial printing field "Name" does not exisat:
`No Name`
Result text if the serial printing field "Name" has the content "John Q. Public":
`John Q. Public`

Explanation:
You can use the non-existence-statement to define what happens if a variable does not exist or is empty.

This statement is especially useful to output serial printing fields and database requests. Please note that the two text statements may include statements again.

Text or Empty Text Statement

`{$Text} / {$}`

The text statement is marked by the control character '$'. The content of the statement is copied directly as a text, i.e. there is no evaluation. This especially useful inside conditional statements. Please not that the text to be copied starts directly after the `$` character . Is the `$` character directly followed by `}`, i.e. the text is empty, then an empty text is copied.

Example 1:
Statement in the text:
`{$This is a text}`

Result text:
`This is a text`

Explanation:
The text is copied without any modification.

Please note that text statements may include statements again.

Comment Statement

`{*Comment}`

The comment statement is not output. This can be used to store hidden information in a drawing.

Example:
Statement in the text:
`{* This is a comment}`

Explanation:
This statement is neither displayed on the screen, nor will it be printed or exported.

Numerical Term Statement, Maximum Number of Decimal Places

`{Term}`

The content of the braces is interpreted as a numerical expression, calculated and output with the maximum number of decimal places (currently 15).

Example:
Statement in the text (the variable “Value“ equals 1):
`{~Value~/3}`

Result text:
`0.333333333333333`

Please note that the decimal separator used depends on user settings, it can be either ‘.’ or ‘,’.

Numerical Term Statement, Rounded

```
{%Term}
```

The content of the braces is interpreted as a numerical expression, calculated, rounded to the active number of decimal places (here five), and then output.

Example:
Statement in the text (the variable “Value“ equals 1):
```
{% ~Value~/3}
```

Result text:
```
0.33333
```

Please note that the decimal separator used depends on user settings, it can be either ‘.’ or ‘,’. The number of decimal places is mostly determined by the active coordinate system. If the statement is part of a dimension text then the number of decimal places specified in the object’s dimension parameters is used. For control texts of the CAM postprocessor the corresponding value from the postprocessor settings is used.

Numerical Term Statement, Rounded, Decimal Point

```
{.Term}
```

The content of the braces is interpreted as a numerical expression, calculated, rounded to the active number of decimal places (here five), and then output. A point is used as decimal separator.

Example:
Statement in the text (the variable “Value“ equals 1):
```
{. ~Value~/3}
```

Result text:
```
0.33333
```

The number of decimal places is mostly determined by the active coordinate system. If the statement is part of a dimension text then the number of decimal places specified in the object’s dimension parameters is used. For control texts of the CAM postprocessor the corresponding value from the postprocessor settings is used.

Numerical Term Statement, Rounded, Decimal Comma

```
{,Term}
```

The content of the braces is interpreted as a numerical expression, calculated, rounded to the active number of decimal places (here five), and then output. A comma is used as decimal separator.

Example:
Statement in the text (the variable “Value“ equals 1):

```
{, ~Value~/3}
```

Result text:

```
0,33333
```

The number of decimal places is mostly determined by the active coordinate system. If the statement is part of a dimension text then the number of decimal places specified in the object’s dimension parameters is used. For control texts of the CAM postprocessor the corresponding value from the postprocessor settings is used.

Numerical Term Statement, Rounded, Decimal Point, Always with Sign

```
{+Term}
```

The content of the braces is interpreted as a numerical expression, calculated, rounded to the active number of decimal places (here five), and then output. A point is used as decimal separator. The number is always preceded with a sign (‘+’ or ‘-’). Zero is always preceded with ‘+’.

Example:
Statement in the text (the variable “Value“ equals 1):

```
{+ ~Value~/3}
```

Result text:

```
+0.33333
```

The number of decimal places is mostly determined by the active coordinate system. If the statement is part of a dimension text then the number of decimal places specified in the object’s dimension parameters is used. For control texts of the CAM postprocessor the corresponding value from the postprocessor settings is used.

Numerical Term Statement, Rounded, Decimal Comma, Always with Sign

`{&Term}`

The content of the braces is interpreted as a numerical expression, calculated, rounded to the active number of decimal places (here five), and then output. A comma is used as decimal separator. The number is always preceded with a sign (‘+’ or ‘-’). Zero is always preceded with ‘+’.

Example:
Statement in the text (the variable “Value“ equals 1):
`{& ~Value~/3}`

Result text:
`+0,33333`

The number of decimal places is mostly determined by the active coordinate system. If the statement is part of a dimension text then the number of decimal places specified in the object’s dimension parameters is used. For control texts of the CAM postprocessor the corresponding value from the postprocessor settings is used.

Numerical Term Statement, Integer

`{#Term}`

The content of the braces is interpreted as a numerical expression, calculated, rounded to zero decimal places (integer), and then output.

Example:
Statement in the text (the variable “Value“ equals 2):
`{& ~Value~/3}`

Result text:
`1`

Floating-Point Value

`{f Term}` or `{fX Term}` or `{f. Term}` or `{f.X Term}` or
`{f, Term}` or `{f,X Term}`

The content of the braces is interpreted as a numerical expression, calculated, and output as floating point number. If the 'f' character is followed by a point or comma, this indicated the decimal separator to be used. If the 'f' character, or the point or comma, respectively, are directly followed by an integer number, this number denotes the number of decimal places to be output. If no number is given, the current default number of decimal places is used.

Example 1:
Statement in the text (the variable "Value" equals 2):
`{f,3 ~Value~/3}`

Result text:
`0,667`

Example 2:
Statement in the text (the variable "Value" equals 2):
`{f.8 ~Value~/3}`

Result text:
`0.66666667`

Exponential Value

`{g Term}` or `{gX Term}` or `{g. Term}` or `{g.X Term}` or
`{g, Term}` or `{g,X Term}`

The content of the braces is interpreted as a numerical expression, calculated, and output as exponential or floating point number (using the shorter version). If the 'g' character is followed by a point or comma, this indicated the decimal separator to be used. If the 'g' character, or the point or comma, respectively, are directly followed by an integer number, this number denotes the number of mantissa places to be output. If no number is given, the current default number of mantissa places is used.

Example 1:
Statement in the text (the variable "Value" equals 2):
`{g,3 ~Value~/3*1e20}`

Result text:
`6,67e+019`

Example 2:
Statement in the text (the variable "Value" equals 2):
`{g.8 ~Value~/3*1e20}`

Result text:
`6.6666667e+019`

Signed Integer Value

`{d Term}` or `{dX Term}`

The content of the braces is interpreted as a numerical expression, calculated, rounded, and output as a signed integer number. If the 'd' character is directly followed by an integer number, this number denotes the number of digits to be output (filled with trailing zeros). If no number is given, the required number of digits is used.

Example 1:
Statement in the text (the variable "Value" equals 2):
`{n ~Value~*600}`

Result text:
`1200`

Example 2:
Statement in the text (the variable "Value" equals 2):
`{n8 ~Value~*600}`

Result text:
`00001200`

Unsigned Hexadecimal Value

`{x Term}` or `{xX Term}`

The content of the braces is interpreted as a numerical expression, calculated, rounded, and output as an unsigned hexadecimal number. If the 'x' character is directly followed by an integer number, this number denotes the number of digits to be output (filled with trailing zeros). If no number is given, the required number of digits is used.

Example 1:
Statement in the text (the variable "Value" equals 2):
`{x ~Value~*600}`

Result text:
`4b0`

Example 2:
Statement in the text (the variable "Value" equals 2):
`{x8 ~Value~*600}`

Result text:
`000004b0`

Unsigned Binary Value

`{b Term}` or `{bX Term}`

The content of the braces is interpreted as a numerical expression, calculated, rounded, and output as an unsigned binary number. If the 'b' character is directly followed by an integer number, this number denotes the number of digits to be output (filled with trailing zeros). If no number is given, the required number of digits is used.

Example 1:
Statement in the text (the variable "Value" equals 2):
`{b ~Value~*600}`

Result text:
`10010110000`

Example 2:
Statement in the text (the variable "Value" equals 2):
`{b16 ~Value~*600}`

Result text:
`0000010010110000`

File Format (MKD)

The following list shows all file elements available in a MKD file of the type “drawing”:

- Scenarios
- Text Styles
- Windows
- Defaults
- Settings
- Plug-In Settings
- Pages
- Custom Colors
- Hatch Types
- Line Sequences
- Coordinate Systems
- Pens
- Line Patterns
- Layers
- Bitmaps
- Variables
- Internal Blocks, Groups, Position Numbers
- Objects and Instances

MKD files of the type “library” or “font”, respectively, do only contain some of the elements listed above.

When opening or saving a file the current **File Options** determine which of these elements are actually loaded or saved respectively. If you want to change the file elements to be loaded or saved only temporarily for the current file operation you can also apply the “Options” button in the **File Selection** dialog.

If you want to import certain file elements into the current drawing from another drawing then choose the command **File > Import > MKD (*.MKD)**. By applying the “Options” button in the **File Selection** dialog you can specify which file elements are to be imported and whether the existing file elements are to be replaced or to be merged with the imported file elements. To export certain file elements use the command **File > Export > MKD (*.MKD)**.

To import or export only the file element “Layers” you can also apply the “Layer Status” button in the **Edit Layers** dialog.

All MKD drawings have the file extension `.MKD` , MKD libraries and fonts have the file extension `.MKL` . These files (if not saved compressed, see **Save Drawing as**) are pure ANSI files, i.e. you can load them into any text editor, like for example Notepad. This can be very helpful if you want to search and replace some character strings in text objects, layer names, pen names, etc. If you want to know more about the MKD file format refer to the **Malz++Kassner Open CAD6 Interface Documentation (MKI_LIB6.HLP)**.

Plug-In Concept (MKI)

Malz++Kassner CAD6 Pro, CAD6 Studio, and CAD6 Industry include the Malz++Kassner Open CAD6 Interface (MKI) - a powerful programming interface that allows the integration of plug-ins (internally sometimes also called modules). These plug-ins can either be import filters, export filters or command extensions in any form. See **Malz++Kassner Open CAD6 Interface Documentation (MKI_LIB6.HLP)** for details.

The Open CAD6 Interface offers an easy way to create powerful extensions to Malz++Kassner applications. The interface was designed to make it as easy as possible to create plug-ins that fit perfectly into the application's environment. Usually, the user won't realize any substantial difference between using an internal command provided by the application or using an external command provided by a plug-in or filter.

Import and export filters can be created with only basic knowledge of the external file format (see **Malz++Kassner Open CAD6 Interface Documentation (MKI_LIB6.HLP)** for documentation on that file format) and some knowledge about geometrical calculations. You will find that especially creating export filters is very easy - if required, the application does all calculations for you and passes simple lines to the export filter.

There are three sample plug-ins including source code and everything else you need to create a plug-in that come with Malz++Kassner CAD6: `IMPORT_.DLL` , `EXPORT_.DLL` , and `SAMPLE_.DLL` . You can use these samples as a basis for your own plug-in development.

After the plug-ins' installation, you can also select the corresponding menu items in Malz++Kassner CAD6 to see how the plug-ins work: **File > Import > Coordinates (*.TXT)**, **File > Export > Coordinates (*.TXT)**, and **Draw > Sample Plug-In >**. For more information on these plug-ins highlight the relevant menu item and press F1.

Chapter 3 - Procedures

Identification

Many commands in Malz++Kassner CAD6 require an object or object element identification.

The object identification works similar for all commands:

- Objects can be identified by simply clicking on their outlines. If several objects lie at the position clicked on, the **Object Choice Dialog** appears above the current mouse position, with the help of which the desired object can be picked out.

Objects in hidden or locked layers usually cannot be identified.

The object identified is sometimes influenced by the selection filter. To activate or deactivate the selection filter please choose the command **Selection Filter (0)** or click the corresponding button in the **Panel**. To edit the parameters of the selection filter please choose the command **Selection Filter Parameters (+0)** or right-click the corresponding button in the Panel.

Object Selection

Many commands in Malz++Kassner CAD6 require an object selection, where the user selects a group of objects that the current command shall be applied to.

The object selection works similar for all commands. Any number of objects can be chosen. This can be done with various methods, which can be used on their own or in combination with one another:

- Objects can be selected by simply clicking on their outlines. If several objects lie at the position clicked on, a small dialog appears above the current mouse position, with the help of which the correct object can be picked out.

- To select several objects quickly, hold down the CTRL key and then click on the outline of each object. If several objects lie at the position clicked on, a small dialog appears above the current mouse position, with the help of which the correct object can be picked out. Release the CTRL key to end the selection.

- To select several object more comfortably, use the **Extended Identification** mode.

- Whole areas can be chosen by holding down the SHIFT key, and specifying two opposite corners of a rectangle. This chooses all the objects which lie wholly within the specified area. Alternately, you can use the **Extra > Identify > Identify Area** command.

Whether only the objects wholly within the area, or also those which overlap it should be chosen is determined by the area mode. This is changed with the **Shift+F6 Key** or the button in the panel.

- All the objects in the drawing can be identified by pressing the **F10 Key**.
- If the same group of objects is to be manipulated several times, it can be selected again by pressing the **F11 Key**.
- To select all but the previously selected object, hold down the STRG key and press **F10 Key** (select all) first, then **F11 Key** (previous selection).
- All the at the moment permanently selected objects can be chosen by pressing the **F12 Key**.

If several of these selection methods are to be combined, the CTRL key must be kept held down throughout the entire selection process. If you then release the CTRL key the current command will be carried out using the selected objects.

All choosing operations are self-inverting. I.e. if an object is chosen twice during a choosing process, it counts as “not chosen”. A third selection re-chooses it, and so on.

Objects in hidden or locked layers cannot be selected.

If all the objects in the drawing except two are to be chosen, follow these steps:

- Press and hold down the CTRL key
- Press the F10 key
- Click, one after another, on the two objects not to be chosen
- Release the CTRL key

The object selection is influenced by the selection filter. To activate or deactivate the selection filter please choose the command **Selection Filter (0)** or click the corresponding button in the **Panel**. To edit the parameters of the selection filter please choose the command **Selection Filter Parameters (+0)** or right-click the corresponding button in the Panel. You can activate, deactivate, or edit the selection filter even during an object selection by keeping the CTRL key pressed until the object selection is completed.

Point Selection

Some commands in Malz++Kassner CAD6 require a point selection, where the user selects a group of definition points that the current command shall be applied to.

The point selection works similar for all commands. Any number of points can be chosen. This can be done with various methods, which can be used on their own or in combination with one another:

- Points can be chosen by simply clicking on them.
- To select several points quickly, hold down the CTRL key and then click on each point. Release the CTRL key to end the selection.
- To select several points more comfortably, use the **Extended Identification** mode.
- Whole areas can be chosen by holding down the SHIFT key, and specifying two opposite corners of a rectangle. This chooses all the points which lie wholly within the specified area. Alternately, you can use the **Extra > Identify > Identify Area** command.
- All the points in the drawing can be chosen by pressing the **F10 Key**.
- If the same group of points is to be manipulated several times, it can be selected again by pressing the **F11 Key**.
- To select all but the previously selected points, hold down the STRG key and press **F10 Key** (select all) first, then **F11 Key** (previous selection).

If several of these selection methods are to be combined, the CTRL key must be kept held down throughout the entire selection process. If you then release the CTRL key the current command will be carried out using the selected points.

All choosing operations are self-inverting, i.e. if a point is chosen twice during a choosing process, it counts as “not chosen”. A third selection re-chooses it, and so on.

Points in hidden or locked layers cannot be selected.

If all points inside an area plus two additional points are to be chosen, follow these steps:

- Press and hold down the CTRL key
- Press and hold down the SHIFT key
- Enter the two corner points of the area
- Release the SHIFT key
- Click, one after another, on the two additional points
- Release the CTRL key

The point selection is influenced by the selection filter. To activate or deactivate the

selection filter please choose the command **Selection Filter (0)** or click the corresponding button in the **Panel**. To edit the parameters of the selection filter please choose the command **Selection Filter Parameters (+0)** or right-click the corresponding button in the Panel. You can activate, deactivate, or edit the selection filter even during a point selection by keeping the CTRL key pressed until the point selection is completed.

Edit Parameters of Current Command

Most commands of Malz++Kassner CAD6 are controlled by a set of parameters that can be edited while the command is active. Some of the commands display a parameter dialog when they are selected from the menu or by any other means, some do not. This depends on whether a command usually requires a new parameter for each usage or not.

In both cases, the command's parameters may be viewed and altered by selecting the **Extras > Change Parameters (+ESC)** command. This command can also be called by pressing the key combination SHIFT+ESC or by clicking the command's icon in the Panel. Unless you have altered the key assignment, you can also press the 'N' key.

Typical commands that do display a parameter dialog on selection:

Draw > Line > Parallel, Numerical
Draw > Line > Angle to Line
Draw > Polyeder > Standard
Draw > Circle > Radius - Object - Object
Trimming > Round Edge, Smooth > Object - Object
Text > Text > Standard
Library > Block > Insert

Typical commands that do not automatically display their parameter dialog, but do still have one:

Draw > Line > Zigzag Line
Draw > Freehand Line
Draw > Hatching > Generated Surface
Trimming > Trim Surface > Generate Surface
Text > Dimension Line > Straight
Text > Dimension > Length, Points
Text > Calculate > Area
Library > Position Number > Assign

Chapter 4 - Screen Elements

Main Window

The Malz++Kassner CAD6 main window serves as a background for the other screen elements. Its main function is to display the menu bar, which provides access to all available commands.

The name of the program, the current zoom factor and of the current drawing are shown In the main window's title bar. If `(UNTITLED)` is shown here, then the drawing is as yet unnamed.

An asterisk after the drawing name (e.g. `MYFILE*`) indicates that the drawing has been changed since it was last saved.

The size of the **Drawing Windows** changes automatically whenever the size of the main window is altered.

System Menu

The main window has a system menu from which certain settings relating to the main window can be changed. Top open it, click onto the icon on the left end of the title bar. In addition to the usual commands for moving and closing the window and altering its size, the following commands are available:

Display Menu Bar

This command determines whether or not the menu bar is displayed. If the menu entry is checked, the menu bar is displayed.

If the menu bar is not displayed, then pressing the ALT key displays a "floating menu" menu at the current mouse position. It can be used like the normal menu bar. Turning the menu bar off has the advantage of increasing the available work space on the screen.

Color Scheme

Use this submenu to determine how the menu bar and all menu entries shall be displayed. The following options are available:

"System Colors"
The menus will be drawn using the standard system colors of the current Windows version.

"Skin Colors"
The menus will be drawn using the colors defined by the currently selected skin (see **Options > Skin**).

"Black on White"
The menus will be drawn black on white for best legibility.

Display Command Icons

If this menu entry is checked, then the associated command icon will be displayed before each menu entry corresponding to a standard command.

Save Position

If this menu entry is checked, then the position of the main window is saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Drawing Window

Malz++Kassner CAD6 can work with up to four drawing windows open at the same time. A different view of the same drawing can be displayed in each window. To do this, a separate coordinate system can be allocated to each drawing window (see **Coordinate Systems > Edit**).

The drawing windows are the program's actual work space. All point entry is carried out in drawing windows. Many commands and keyboard commands are only available when the mouse pointer is within a drawing window.

The mouse position in a drawing window is indicated by a cross hair and not by a pointer. The crosshair makes it easier to recognize horizontal and vertical movement.

The intersection of the crosshair's lines is the current drawing position. The coordinates of this point are shown in the Status Line if it is turned on. In addition, the mouse position can be read from both Rulers.

The number of drawing windows is controlled with the command **Options > Windows**. Several layouts are available which all have one thing in common. All drawing windows in a layout have the same height/width relationship. This has the advantage that the contents of two windows can easily be swapped. (e.g. see **Enlarge Window 2**).

The size of the drawing windows changes automatically if the size of the main window is altered. Individual drawing windows cannot be resized.

If several drawing windows are visible, then one of them will always be “active”. Which window is active can be seen by the way the rulers are displayed. A window is activated by clicking on its rulers with the left mouse button. Many commands which modify window-specific settings (e.g. **Zoom > Section**) always relate to the active window.

If only one drawing window is displayed, it is automatically the active window.

Context Menu

The drawing window has a context menu from which certain settings relating to it can be changed. Right-click the window's scale to open it. The following commands are available (partly depending on the clicked position):

Select Coordinate System

Use this submenu to select the active coordinate system for this drawing window (see **View > Coordinate Systems > List**).

Edit Element

Use this command to edit the elements associated to this button (layers, pens, coordinate systems, hatchings, or pages) in the appearing dialog.

Display

Use this submenu to determine for which of the five elements (layers, pens, coordinate systems, hatchings, or pages) a selection button shall be displayed along with an insert of the element's name next to it.

Opacity

Use this submenu to determine the opacity of the element names' insert, i.e. how much the drawing area shall be concealed.

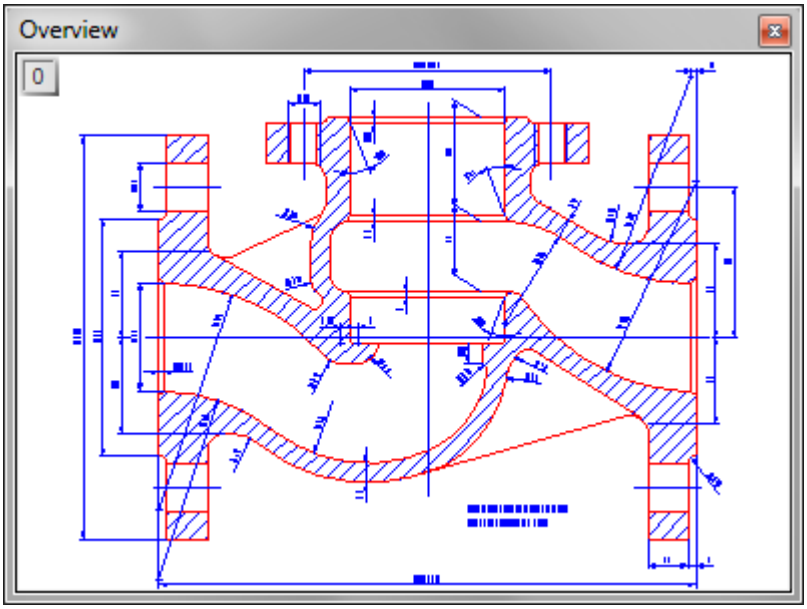
Workflow Manager

The Workflow Manager is a context-sensitive menu that usually pops up if you click the right mouse button. It offers a selection of commands that are useful at the moment.

To allow a fast workflow, the most important command is always arranged closest to the mouse cursor. If necessary, the order of menu items is reversed to display the most important command at the bottom of the menu.

Once the workflow manager is open, a right mouse click will display either the standard menu or the **Popup Menu** to select further commands.

Overview Window



Basically, the overview window offers the same possibilities than a “normal” **drawing window** does. But in addition to that, the overview window can be scaled and placed anywhere on screen, and it can be configured to use a simplified display that leaves out or simplifies some object types. Finally, the automatic window update after each operation can also be disabled in order to save time.

Context Menu

The overview window has a context menu from which certain settings relating to it can be changed. Right-click the window's scale to open it. Apart from the usual commands of all drawing windows, the following additional commands are available:

Automatic Update

Only if this menu entry is checked, the overview window will automatically be updated when changes in the drawing occur. Otherwise, you will have to update it manually by moving the mouse above the overview window and pressing the SPACE key.

Simplified Display

If this menu entry is checked, the overview window will show a simplified view of the drawing with minimum line widths, only solid lines, no markings, no rulers (even if active in other windows), etc.

Visible

Use this command to hide the window. To show it again, use **Window > Overview on/off**.

Rearrange

Pushes the window into one of the four corners of the workspace. The overview window is normally in the upper right corner of the drawing window.

Save Position

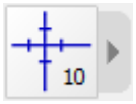
If this menu entry is checked, then the window position will be saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Panel

The panel is located on the left of the Malz++Kassner CAD6 main window. It is made up of many buttons with which various functions can be activated. In addition, the buttons show the status of the corresponding functions.

In general the setting can be changed by clicking on the button with the left mouse button. Clicking into the window with the right mouse button will display the context menu.

The panel consists of the following elements:



Snapping functions and Snapping radius
(see **Snap Modes > Snap Active (F6)** and **Snap Modes > Radius**)

Snap Mode "Midpoint" (see **Snap Modes > Center**)

Snap Mode "Quadrant" (see **Snap Modes > Quadrant**)

Snap Mode "Edge" (see **Snap Modes > Edge**)

Snap Mode "Corner / End-Point" (see **Snap Modes > Corner / End-Point**)

Snap Mode "Intersection" (see **Snap Modes > Intersection**)

Snap Mode "Construction Aid" (see **Snap Modes > Construction Aid**)

Snap Mode "Marking" (see **Snap Modes > Marking**)

Snap Mode "Other Point" (see **Snap Modes > Other Point**)

Snap Mode "Relative" (see **Snap Modes > Relative**)

A collection of the most important input parameters:

- Position Grid (see **Coordinate Systems > Position Grid Active (F4)**)
- Orthogonal Mode (see **Orthogonal Mode (F5)**)
- Area Mode (see **Toggle Area Mode (+F6)**)
- Duplicate / Multiline (see **Duplicate / Multiline (F7)**)
- Selection Filter (see **Selection Filter (0)**)

- Extended Identification (see **Identify > Extended Identification on/off**)
- Arc Mode (see **Arc Mode > Toggle Arc Mode (+F2)**)
- Arc Direction (see **Arc Mode > Toggle Arc Direction (F2)**)

A collection of the most important display settings:

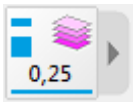
- Display Grid (see **Coordinate Systems > Display Grid Active (F3)**)
- Definition Point Display (see **Drawing Settings > Screen**)
- Toggle Display Mode for hidden layers of the current page (see **Layer Display > Display All Layers Normally**)
- Toggle Display Mode for objects assigned to non-active pages (see **Page Display > Classic (All)**)
- Construction Aid Display (see **Display Construction Aid (F9)**)
- Construction Aid Locking (**see Lock Construction Aid (+F9)**)
- Comment Display (see **Comment > Display**)

Current command's icon (without indicator)

Current command's icon (no parameters available)

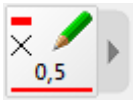
Current command's icon (parameters are available, see **Edit Parameters**)

Selection button for layers (without preview)



Selection button for layers (with preview)

Selection button for pens (without preview)



Selection button for pens (with preview)

Selection button for coordinate systems

Selection button for hatching types

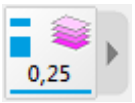
Selection button for pages (current page number in the top row, total number of pages in the bottom row)

Selection button for scenarios and further scenario commands

“Joystick” button for window movement

Layer Button Preview

If the preview is active, the layer selection button displays a lot of information simultaneously:



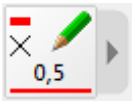
The upper left color field shows an inherited line color for screen display. If the filling mode is either “Filling” or “Eraser”, i.e. no outline is drawn, this field will show only a cross.

The lower left color field shows an inherited fill color for screen display. If the filling mode is ”Outline”, i.e. no filling is drawn, this field will show only a cross.

Below the two color fields, an inherited line width is displayed in millimeters. At the bottom, the inherited line pattern is displayed. Both the line width and line pattern display are based on the properties for output.

Pen Button Preview

If the preview is active, the pen selection button displays a lot of information simultaneously:



The upper left color field shows the line color for screen display. If the filling mode is either ”Filling” or ”Eraser”, i.e. no outline is drawn, this field will show only a cross.

The lower left color field shows the fill color for screen display. If the filling mode is ”Outline”, i.e. no filling is drawn, this field will show only a cross.

Below the two color fields, the line width is displayed in millimeters. At the bottom, the line pattern is displayed. Both the line width and line pattern display are based on the properties for output.

"Joystick" Button

Click on the button's edges for one movement step at time, or click and hold the button center to scroll fluidly.

The scrolling speed varies depending of how far the cursor is moved away from the button's center. While atop the "joystick" button, the mouse wheels will work for zooming and/or moving.

Context Menu

The panel window has a context menu from which certain settings relating to it can be changed. Right-click inside the window to open it. The following commands are available (partly depending on the clicked position):

Display Parameters Dialog

Use this command to edit the parameters associated to the clicked button.

Edit Element

This command can be used to edit the element associated to this button (layer or pen) in the appearing dialog.

Edit Button Assignment

This command can be used to edit the tool button's assignment by choosing the desired command in the appearing graphic menu.

Insert Button

This command can be used to insert a new button before the clicked one by choosing the desired command in the appearing graphic menu. All following buttons will be moved down one step.

Move Button to Begin

This command can be used to move the clicked button to the beginning of the button list.

Move Button One Upward

This command can be used to move the clicked button one step upward.

Move Button One Downward

This command can be used to move the clicked button one step downward.

Move Button to End

This command can be used to move the clicked button to the end of the button list.

Delete Button

This command can be used to delete the clicked button. All following buttons will be moved down one step.

Reset All Buttons

This command can be used to reset all button to their respective default assignment.

Display

This submenu can be used to determine which buttons shall be displayed in the panel.

Opacity

This submenu can be used to determine the opacity of the element names' insert, i.e. how much the drawing area shall be concealed.

Visible

Use this command to hide the window. To show it again, use **Window > Panel on/off**.

Placeable

If this menu entry is checked, then the window can be placed anywhere on the screen. Otherwise, it will be aligned to the left edge of the screen.

Rearrange

Pushes the window into one of the four corners of the workspace. The panel window is normally in the upper left corner of the drawing window.

Save Position

If this menu entry is checked, then the window position will be saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

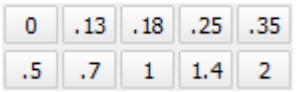
Command Bar

The command bar contains several groups of buttons, each of which containing a useful set of related commands and settings. Those button group can be switched on and off individually. Furthermore, additional buttons and separators can be added.

Click with the right mouse button into the window to display the context menu in which to determine (among other settings) the button groups that shall be displayed in the command bar.

Some buttons groups are used to alter properties. During standard operation, these buttons will directly affect the current properties of pen "*Standard". When drawing without the use of pens, the command bar offers the fastest way to select the desired properties.

This button is used to select a line pattern. When left-clicking it a list of all currently defined line patterns appears.



These buttons are used to select the line width. They do always display the line width in millimeters (independent of the current unit setting)! Clicking with the left mouse button selects the line width, clicking with the right mouse button displays a dialog window where the line width assigned to that button can be edited.

These buttons are used to select the line color. Clicking with the left mouse button selects the line color, clicking with the right mouse button displays a dialog window where the line color assigned to that button can be edited.

These buttons are used to set the filling mode. The available filling modes are (in this order): "Outline", "Filling", "Filling & Outline", "Eraser", "Eraser & Outline".

These buttons are used to select the fill color. Clicking with the left mouse button selects the fill color, clicking with the right mouse button displays a dialog window where the fill color assigned to that button can be edited.

During the execution of the command **Modify Objects**, the command bar works differently.

Pressing property buttons in the command bar will now directly alter the properties of the currently selected objects! Since multiple objects with different properties might be selected, the property button of the command bar will not be displayed permanently pressed. This would lead to the incorrect assumption that the button state visualizes the objects properties, which is not the case.

Context Menu

The command bar has a context menu from which certain settings relating to it can be changed. Right-click inside the window to open it. The following commands are available (partly depending on the clicked position):

Insert Button Group

This command can be used to insert an additional button group left to the clicked button group. When inserting plain buttons, those will initially be undefined.

Edit Button Assignment

This command can be used to edit the tool button's assignment by choosing the desired command in the appearing menu (depending on the choice, either the menu, a layer list or a pen list).

Move Button Group to Begin

This command can be used to move the clicked button group to the beginning of the button list.

Move Button Group One Leftward

This command can be used to move the clicked button group one step leftward.

Move Button Group One Rightward

This command can be used to move the clicked button group one step rightward.

Move Button Group to End

This command can be used to move the clicked button group to the end of the button list.

Delete Button Group

This command can be used to delete the clicked button group. All following button groups will be moved left one step.

Reset All Button Groups

This command can be used to reset the command bar to its default assignment.

Visible

Use this command to hide the window. To show it again, use **Window > Command Bar on/off**.

Placeable

If this menu entry is checked, then the window can be placed anywhere on the screen. Otherwise, it will be aligned to the upper edge of the screen.

Rearrange

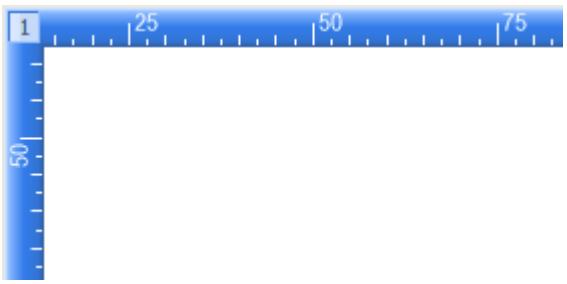
Pushes the window into one of the four corners of the workspace. The command bar is normally in the upper left corner of the drawing window.

Save Position

If this menu entry is checked, then the window position will be saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Ruler

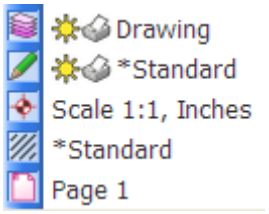
Usually each drawing window has a ruler, which runs along the left and upper edge of the window. It fulfils several purposes.



Above all, it serves the purpose of orientation when working with coordinates. You can read from the ruler the current position of the origin and the coordinates of the crosshair.

When working with more than one drawing window, the ruler also shows which window is "active". The ruler of the active window is displayed normally, but the ruler of an inactive window is shown, like unavailable buttons, in a different color. In the upper left corner of each window at the rulers' intersection, the number of the window is shown.

A drawing window can be activated by clicking on its ruler with the left mouse button. Clicking with the right mouse button calls the context menu (see **Drawing Window**).



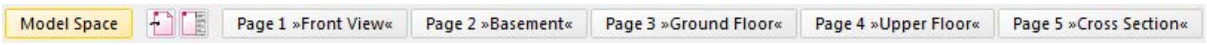
At the lower end of the left ruler, some buttons for direct element selection (layers, pens, coordinate systems, hatchings, and pages) are displayed unless they were deactivated. Clicking them with the left mouse button displays the corresponding selection menu.

If ruler display is turned off using **Options > Windows**, a small field in the upper left corner of the window remains active. This field shows which window is active. Clicking on this field with the left or right mouse button has the same effect as clicking on the ruler.

The window number is shown in this field.

Layout Window

The layout window will be displayed automatically once the page display mode is set to layout mode (see **View > Page Display > Layout (Active Page / Model)**).



It contains several buttons for fast switching between model space and defined pages.

Status Window

The status window can be placed anywhere on the screen. It has two functions: to display coordinates and other details for the current operation, and the direct entry of coordinates.

Its appearance depends upon where the mouse pointer is, and at what stage a command is.

The currently active command's icon is always shown at the left of the status window. The expected input is shown in the title bar.

Clicking the mouse button in the status window calls direct coordinate entry (see **Coordinate Entry (F8)**).

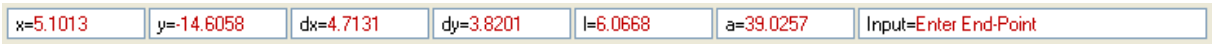
The way in which the status window functions depends on the position of the mouse pointer or crosshair and the stage of command input.

Inside a Drawing Window

Point Entry without Reference Point

"x" and "y" Current crosshair position
"Used" Used portion of currently reserved memory in KB (1024 bytes)
"Command" Description of the current command
"Input" Description of the currently expected input

Point Entry with Reference Point



"x" and "y" Current crosshair position
"dx" and "dy" Difference between the current crosshair position and the position of the reference point
"l" Distance of the current crosshair position to the reference point
"a" Angle of the current crosshair position relative to the reference point
"Input" Description of the currently expected input

Hovering a button

“Button” Description of the button under the mouse pointer
“Input” Description of the currently expected input

Context Menu

The status window has a context menu from which certain settings relating to it can be changed. Right-click inside the window to open it. The following commands are available (partly depending on the clicked position):

Visible

Use this command to hide the window. To show it again, use **Window > Status on/off**.

Placeable

If this menu entry is checked, then the window can be placed anywhere on the screen. Otherwise, it will be aligned to the lower edge of the screen.

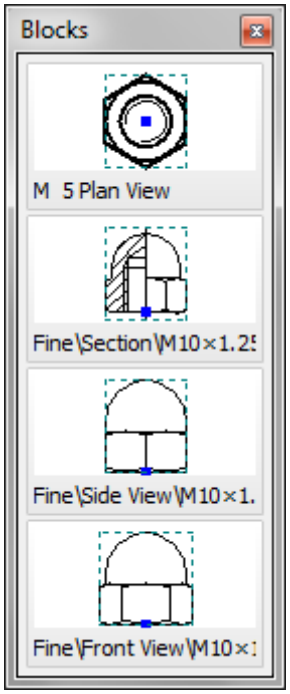
Rearrange

Pushes the window into one of the four corners of the workspace. The status window is normally in the lower left of the drawing window.

Save Position

If this menu entry is checked, then the window position will be saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Block List



The block list contains 200 buttons which can have blocks allocated to them. This makes it easy to access frequently used blocks rapidly.

Left-clicking on a button allows the block associated with that button to be placed in the drawing. It hangs from the crosshair and drops into the drawing at the position of the crosshair when the left mouse button is clicked again. This operation can be repeated until the right mouse button is clicked, ending the command sequence.

Right-clicking allows the button allocation to be changed. It calls the Insert Block dialog, in which the library and block name as well as other parameters like scaling and rotation can be specified. In addition, the button's block allocation can be cleared.

The values specified here are saved separately for each button, so the same block may be saved several times with different parameters (for example, rotated by 0° - 90° - 180° - 270°).

Context Menu

The block list has a context menu from which certain settings relating to it can be changed. Right-click inside the window to open it. The following commands are available (partly depending on the clicked position):

Edit Button Assignment

This command can be used to edit the tool button's assignment by choosing the desired command in the appearing Insert Block dialog.

Insert Button

This command can be used to insert a new button before the clicked one by choosing the desired command in the appearing Insert Block dialog. All following buttons will be moved down one step.

Move Button to Begin

This command can be used to move the clicked button to the beginning of the button list.

Move Button One Upward

This command can be used to move the clicked button one step upward.

Move Button One Downward

This command can be used to move the clicked button one step downward.

Move Button to End

This command can be used to move the clicked button to the end of the button list.

Delete Button

This command can be used to delete the clicked button. All following buttons will be moved down one step.

Reset All Buttons

This command can be used to delete all button assignment.

Number of Columns

This submenu can be used to determine the number of permanently displayed button columns.

Additional Dynamics

This submenu can be used to determine the number of additional, dynamically displayed button columns, i.e. columns that pop up if the mouse pointer is within the window.

Button Width

This submenu can be used to determine the width of the buttons.

Display

This submenu can be used to determine which information shall be displayed in the buttons.

Sort Buttons

The commands in this submenu can be used to sort the buttons in different modes.

Visible

Use this command to hide the window. To show it again, use **Window > Block List on/off**.

Placeable

If this menu entry is checked, then the window can be placed anywhere on the screen. Otherwise, it will be aligned to the left edge of the screen.

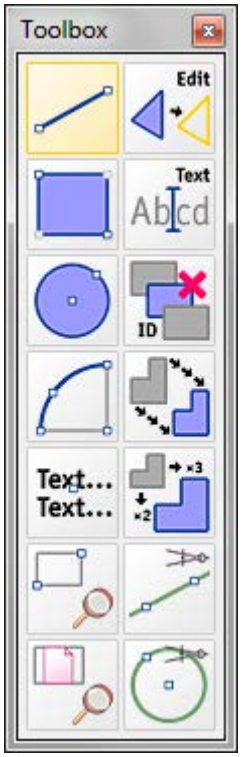
Rearrange

Pushes the window into one of the four corners of the workspace. The block list is normally in the upper left corner of the drawing window.

Save Position

If this menu entry is checked, then the window position will be saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Toolbox



The toolbox window contains up to 1,000 buttons which can have program commands assigned to them. This enables frequently used commands to be accessed easily.

Clicking briefly on one of the buttons with the left mouse button chooses the command associated with that button.

Clicking and holding down (for 1/2 second or more) the left mouse button on a tool window button calls a sub menu. This submenu contains all the commands from the same submenu as the one the button's command is located in. Another command can be chosen from this submenu; this command will be allocated to the button and immediately carried out.

This only works, naturally enough, for commands which appear in a submenu. If the command appears in a menu, holding down the mouse button has no effect and the command is carried out immediately.

Context Menu

The panel window has a context menu from which certain settings relating to it can be changed. Right-click inside the window to open it. The following commands are available (partly depending on the clicked position):

Edit Button Assignment

This command can be used to edit the tool button's assignment by choosing the desired command in the appearing graphic menu.

Insert Button

This command can be used to insert a new button before the clicked one by choosing the desired command in the appearing graphic menu. All following buttons will be moved down one step.

Move Button to Begin

This command can be used to move the clicked button to the beginning of the button list.

Move Button One Upward

This command can be used to move the clicked button one step upward.

Move Button One Downward

This command can be used to move the clicked button one step downward.

Move Button to End

This command can be used to move the clicked button to the end of the button list.

Delete Button

This command can be used to delete the clicked button. All following buttons will be moved down one step.

Reset All Buttons

This command can be used to reset all button to their respective default assignment.

Number of Columns

This submenu can be used to determine the number of permanently displayed button columns.

Additional Dynamics

This submenu can be used to determine the number of additional, dynamically displayed button columns, i.e. columns that pop up if the mouse pointer is within the window.

Small Icons

If this menu entry is checked, small icons (24 × 24 pixels) will be displayed in the button. Otherwise, standard icons (40 × 40 pixels) will be displayed.

Opacity

This submenu can be used to determine the opacity of the element names' insert, i.e. how much the drawing area shall be concealed.

Visible

Use this command to hide the window. To show it again, use **Window > Toolbox on/off**.

Placeable

If this menu entry is checked, then the window can be placed anywhere on the screen. Otherwise, it will be aligned to the left edge of the screen.

Window Position

This submenu can be used to determine the window's position (left or right to the drawing area).

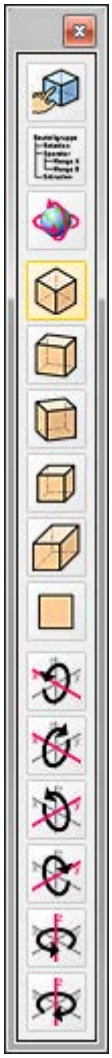
Rearrange

Pushes the window into one of the four corners of the workspace. The toolbox window is normally in the upper left corner of the drawing window.

Save Position

If this menu entry is checked, then the window position will be saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Plug-In Window



The plug-in window contains up to 100 buttons controlled by plug-ins.

Clicking briefly on one of the buttons with the left mouse button chooses the plug-in function associated with that button.

If applicable, clicking the right mouse button on a plug-in window button calls an options dialog window associated with this button. The exact behavior of the button is defined by the calling application.

Context Menu

The plug-in window has a context menu from which certain settings relating to it can be changed. Right-click inside the window to open it. The following commands are available (partly depending on the clicked position):

Display Parameters Dialog

Use this command to edit the parameters associated to the clicked button.

Visible

Use this command to hide the window. To show it again, use **Window > Plug-In Window on/off**.

Placeable

If this menu entry is checked, then the window can be placed anywhere on the screen. Otherwise, it will be aligned to the right edge of the screen.

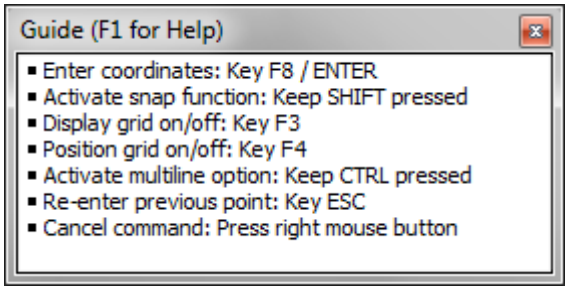
Rearrange

Pushes the window into one of the four corners of the workspace. The plug-in window is normally in the upper right corner of the drawing window.

Save Position

If this menu entry is checked, then the window position will be saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Guide Window


Guide (F1 for Help)
Enter coordinates: Key F8 / ENTER
Activate snap function: Keep SHIFT pressed
Display grid on/off: Key F3
Position grid on/off: Key F4
Activate multiline option: Keep CTRL pressed
Re-enter previous point: Key ESC
Cancel command: Press right mouse button

The guide window displays some hints based on the currently active command and the expected point entry. They show you all available options, how you can alter settings that influence the current command (especially the key sequences), and what the application is expecting from you.

Context Menu

The plug-in window has a context menu from which certain settings relating to it can be changed. Right-click inside the window to open it. The following commands are available (partly depending on the clicked position):

Visible

Use this command to hide the window. To show it again, use **Window > Guide on/off**.

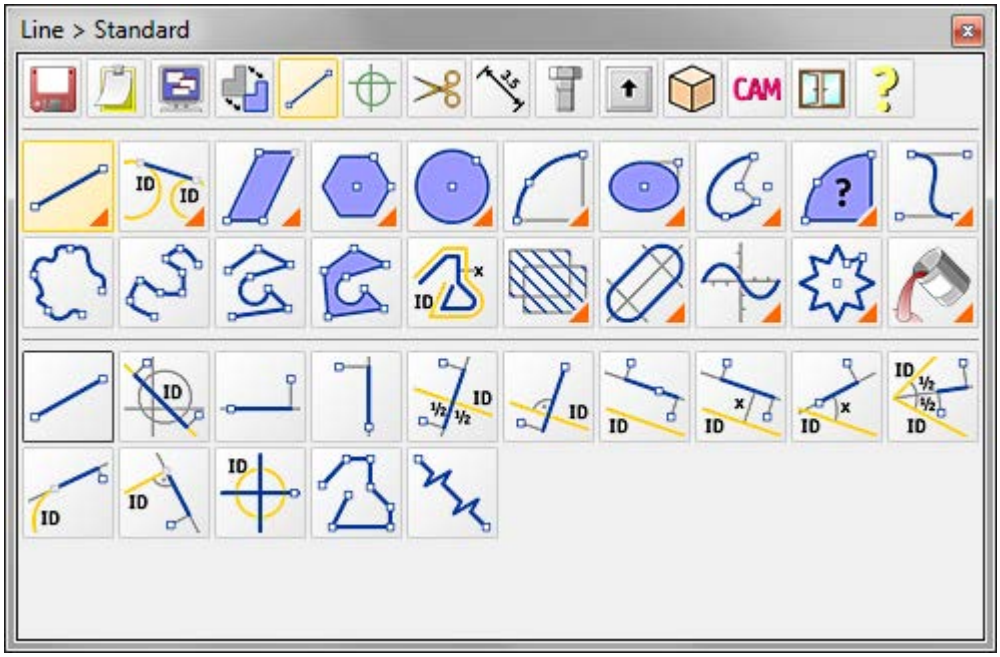
Rearrange

Pushes the window into one of the four corners of the workspace. The guide window is normally in the lower right corner of the drawing window.

Save Position

If this menu entry is checked, then the window position will be saved when leaving the program. The next time that the program is started, it appears in the same place on the screen. The position is saved relative to the whole screen.

Popup Menu



The popup menu is a graphical menu which uses icons to represent commands instead of text descriptions. It always appears at the position of the mouse pointer. The popup menu is structured in the same way as the normal menu; it has the same menus, menu entries, and submenus as the normal menu.

Choosing a command can follow the same procedure as using the text menus. Firstly, the menu is chosen from the upper area; the menu entries and submenus from that menu appear in the central area. Choosing a menu entry carries out that command, or in the case of a submenu entry, the commands from the submenu appear in the lower area.

Submenu icons have small solid orange triangle in the lower right corner of submenu icons to help distinguish them. Submenu icons always show the “most important” command from that submenu.

A command, menu or submenu is chosen by clicking on its icon with the left mouse button. The choice of command can be cancelled by pressing ESC or clicking the right mouse button in the Popup Menu.

The title bar of the Popup Menu shows the description of the command or menu under the mouse pointer. Help on this command can be called by pressing the **Key F1**.

The icons used in the Popup Menu are the same as those used in the **Toolbox** and to display the current command in the **Status Line**.

Chapter 5 - Examples

Rules

Before you start

The first part of this chapter explains, step by step, how to make a drawing. A requirement to follow this example through is an identical system configuration. Therefore we recommend that you import the file elements “Page Format”, “Line Sequences”, “Coordinate Systems”, “Pens”, “Line Patterns”, and “Layers” from the `Tutorial.mkd` configuration file by means of the command **File > Import > Malz++Kassner CAD6 (*.MKD)**. If other settings are needed, this is made clear at the start of the example.

The file `Tutorial.mkd` along with other sample files is located in the `Tutorial` directory in your Malz++Kassner CAD6 directory. In case you have downloaded the tutorial from our Web site then you should copy the downloaded drawing files (`*.MKD`) into the `Tutorial` directory in your Malz++Kassner CAD6 directory. The file `MKTUTOR6.CHM` should be always located directly in the Malz++Kassner CAD6 directory.

Standards

All of these examples are based on DIN 15 (DIN = Deutsche Industrienorm, German Standard). This standard lays down, amongst other things, which line type is used for which purpose. In technical drawing, 2 line groups are used most of the time, identified by their full line widths. These are the line groups 0.5 and 0.7. Only lines from *one* line group should be used in each drawing.

The choice of line group should be made according to the type of part to be drawn. Simple, large parts should be drawn using the 0.7 group and smaller more detailed ones with the 0.5 group.

Representation

In the examples, the Construction Aid is shown as a dotted line. Drawn objects are shown with the relevant pen properties.

Defaults

For certain object types (dimensions, dimension numbers and text) it is possible to specify the pens to be used before starting the drawing. These defaults are set by choosing **View > Pens > Defaults**. If the line group 0.5 is usually used, the pen “0.5 mm\Solid Line Narrow” should be assigned to these object types as the default. This saves the need to change pens when adding a dimension; the program does it automatically.

The Rectangle

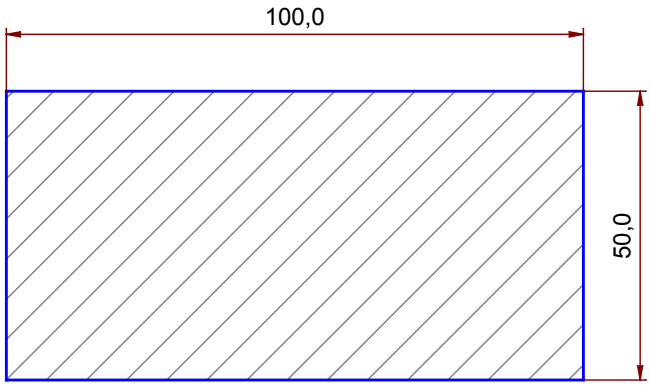


Figure 1: The Rectangle

Aim

A rectangle of fixed size is to be drawn, measured, hatched and centered on the page.

Settings

`Tutorial.mkd`

Firstly, draw the rectangle to size. Choose the pen “0.5 mm\Solid Line Wide”, which has the correct set of properties (solid line, 0.5 mm) for object edges.

Choose **Polygon > Rectangle** from the **Draw** menu. The status line gives you information about the progress of the command. Follow the prompt “Enter Corner 1” and click the left mouse button at the upper left of the page. It can be released immediately. The position of this point is not really important as the rectangle is to be centered on the page anyway. This mouse click has provisionally marked the upper left corner of the rectangle.

The status line now prompts you to enter the second corner point. As this point is to be at a precisely determined x and y distance (also called a relative distance) from the first corner point, enter this distance with the keyboard by pressing the ENTER key. In the appearing dialog, a field with an insertion mark appears. Move the insertion mark to the dx field of the dialog by pressing the TAB key multiple times, type `100` , move the mark to the next field, type `-50` and press ENTER again.

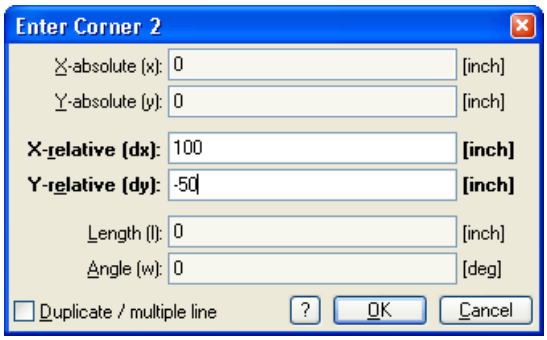


Figure 2: Direct Input

This tells the program to enter the second corner point 100 mm in the x direction and -50 mm in the y direction from the first corner point. Now you have a rectangle of the right dimensions (Figure 3). The dimensions can be checked by measuring the object.

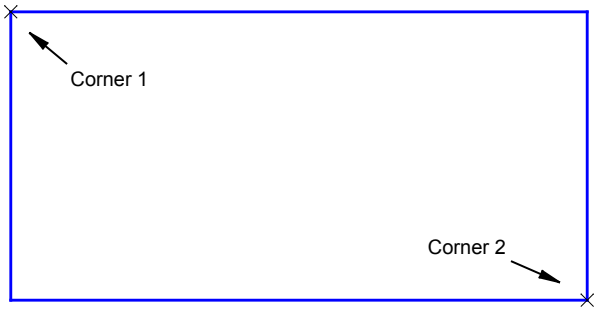


Figure 3

If you have already chosen a pen for dimensions using **View > Pens > Defaults** it is not necessary to change pens. If this is not the case, choose the pen "0.25 mm\Solid Line Narrow" from the panel.

Choose the command **Dimension > Length, Object** from the **Lettering** menu. The dialog contains dimension parameters for most situations. Confirm these settings by clicking on "OK". The status line now prompts you to identify a reference line. This line is the line which is to be measured. Click on the upper horizontal line of the rectangle with the left mouse button. This identifies the line. Next, specify the dimension line position by moving the mouse vertically (perpendicular to the reference line). You will see that the dimension line always moves in fixed increments. Click the left mouse button when the line has jumped one position upwards. This completes the first measurement.

Without re-starting the command (it is still active) you can measure the right (vertical) side of the rectangle in the same way (Figure 4).

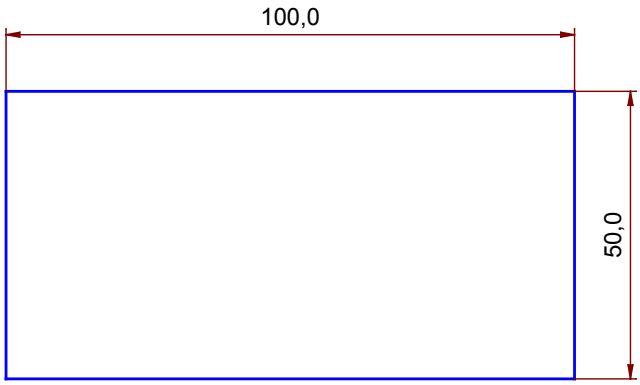


Figure 4

Next, apply a hatching to the rectangle. Activate the pen “0.25 mm\Solid Line Narrow” and choose the **Hatching > Objects** command from the **Drawing** menu. To apply the hatching, identify the rectangle by clicking on its edge (Figure 5).

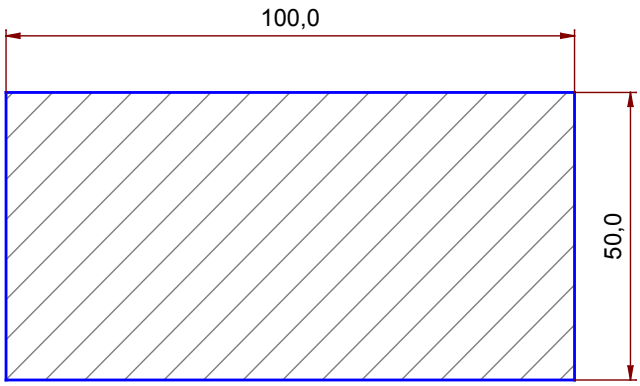
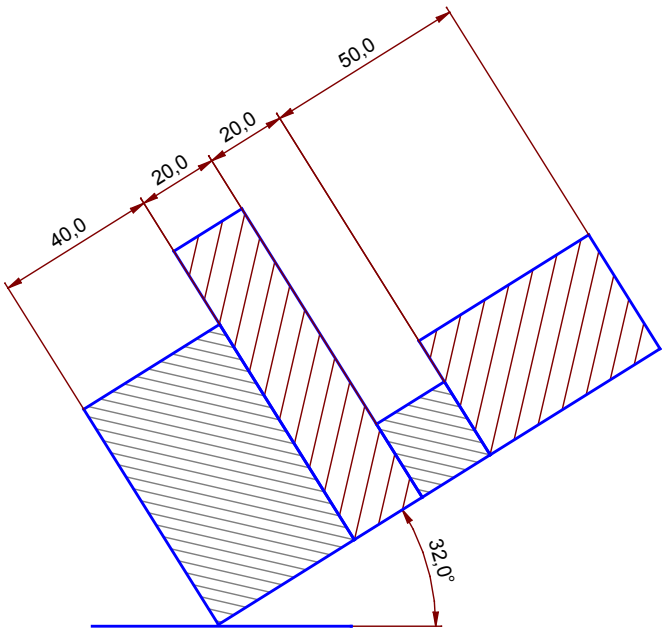


Figure 5

Finally, choose **Shape > Align Objects > Centered, Page Both** and press F10 to center the whole drawing relative to the page margins. Choosing **View > Zoom > Page** allows you to see the effect properly.

To save your first drawing, choose **File > Save As**, enter a file name, e.g. `Drawing1.mkd`, and click on “OK”. You can then enter a full description of the drawing in a further dialog and carry out the save by clicking on “OK”.

Snapping Functions and Compound Measurements



This chapter uses some commands that are only available in Malz++Kassner CAD6 Pro, CAD6 Studio, and/or CAD6 Industry!

Aim

The following example shows ways in which the Construction Aid and snapping functions can be used and in addition how to set dimension parameters to cope with making complicated measurements.

Settings

`Tutorial.mkd`

Procedure

First of all, the rectangles are drawn and hatched, then rotated and measured.

This drawing does not contain any information about the height of the rectangles. You can choose any height you want as the principle remains the same. However, it is necessary to stick to a predetermined width and this is done with the help of the Construction Aid. Use the command **Construction > Construction Aid Endless Line > Vertical** to place a first vertical Construction Aid line at the left of your page.

Use the command **Construction > Construction Aid Endless Line > Parallel, Numerical** to insert another vertical line parallel to the existing one. After choosing the command, a dialog appears in which you can enter the required spacing. Enter 40 mm (the width of the first rectangle) and identify the previously drawn vertical. Move the crosshair to the right until the red line separates from the identified line and jumps 40 mm to the right. Confirm

this by clicking the left mouse button and then end the command sequence (click right mouse button, and if the Workflow Manager is active and appears, choose **Finish Command**).

Because the parallel lines are placed in the correct positions automatically, snapping must remain turned off during this command. Otherwise, conflicts can arise between the different positioning techniques. If the result of the poisoning leads to a conflict, the loudspeaker beeps.

To reactivate the **Construction > Construction Aid Endless Line > Parallel, Numerical** command, press SHIFT and ESC at the same time, then press the “Options” button. The same dialog as before appears and you can set the value 20mm for the second line’s spacing. Proceed as previously: identify the last line and move the crosshair until the red line jumps 20 mm to the right. Click the mouse button to confirm this and move the crosshair again until the line jumps another 20 mm, click the left mouse button to confirm the command. Then end the command sequence (click right mouse button, and if the Workflow Manager is active and appears, choose **Finish Command**).

After pressing SHIFT+ESC and the button “Options” again, enter the last interval of 50 mm and proceed as previously.

Figure 1: Construction Aid

To finish constructing the Construction Aid use the command **Construction > Construction Aid Endless Line > Horizontal** to place a horizontal line in the lower part of the page. The Construction Aid should resemble Figure 1.

To draw the first rectangle, activate snapping and the “Construction Aid” snapping mode. Choose **Draw > Polygon > Rectangle** and click on the intersection of the left Construction Aid line with the horizontal Construction Aid line (Point 1, Figure 2).

Because you can choose the height of the rectangle freely and only have to place the rectangles second corner on the second vertical line, activate the “Edge” snapping mode and then place the second corner point on the vertical Construction Aid line which you previously placed exactly 40 mm to the right. (Point 2, Figure 2).

Use a similar method to draw the second, third and fourth rectangles (Figure 2).

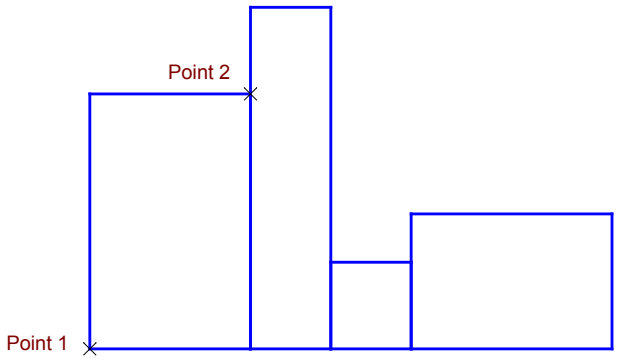


Figure 2: Basic elements

Next, hatch the rectangles. Apart form the standard hatching in rectangles 2 and 4, it’s necessary to define a narrower -45° hatching for rectangles 1 and 3.

First, choose **Draw > Hatching > Edit Line Sequences** and create a new line sequence by clicking on the “New” button. Enter `Solid Line 2.5 mm` , click on “OK” and then click on the “Edit” button to edit this line sequence and set all distances to 2.5 mm. Close both dialogs by clicking “OK”.

Now, choose **Draw > Hatching > Edit Hatching Types** and create a new hatching type by clicking on the “New” button. Enter `Hatching Two` , click on “OK” and then click on the “Edit” button to specify the new hatching’s properties. The only alterations needed are to change the first line sequence to “Solid Line, 2.5 mm” and the angle to -45°. Close the top-level dialog by clicking on “OK”, then the main dialog by clicking on “Activate”.

The new hatching becomes the current hatching. Choose **Draw > Hatching > Objects** and click on the first and third rectangle (Figure 3).

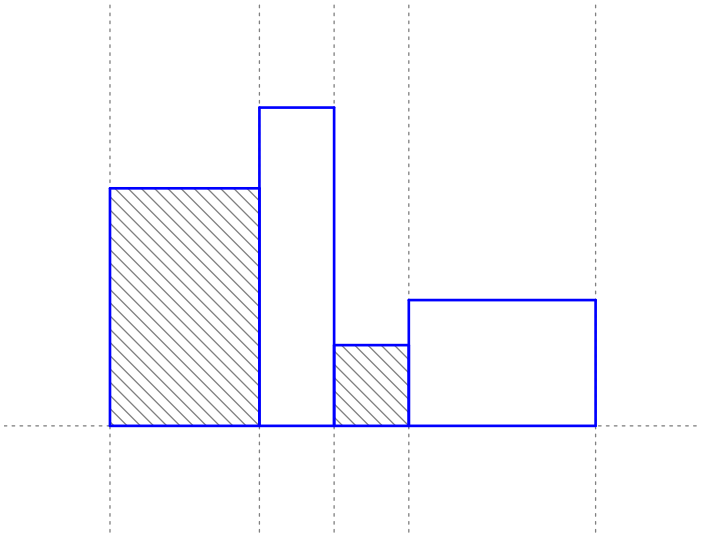


Figure 3: First hatching

To re-select standard hatching, choose **Draw > Hatching > Hatching Type List** and select "*Standard" hatching type from the list which appears.

Rectangles 2 and 4 can now also be hatched easily by choosing **Draw > Hatching > Objects** (also available from the command list, simply press the key H) and clicking on them (Figure 4).

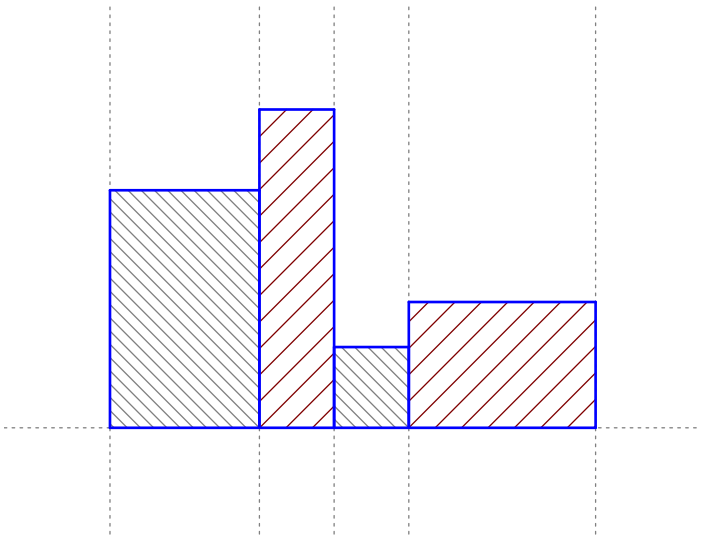


Figure 4: Second hatching

Next, all the objects are to be rotated by 32°. To do this, choose **Shape > Rotate Objects > Center**. and identify the objects to be rotated. In this case all the objects on the page are to be rotated, and it is easiest to identify them all by using the function key F10 (or the keyboard combination SHIFT+Q). Alternatively, you can hold down the SHIFT key and drag a frame around all the objects. Enter “32” into the dialog which appears and click on “OK” (Figure 5).

As the Construction Aid is not required any more, it can be turned off by pressing the F9 key or clicking on the relevant button in the panel. This makes the drawing clearer.

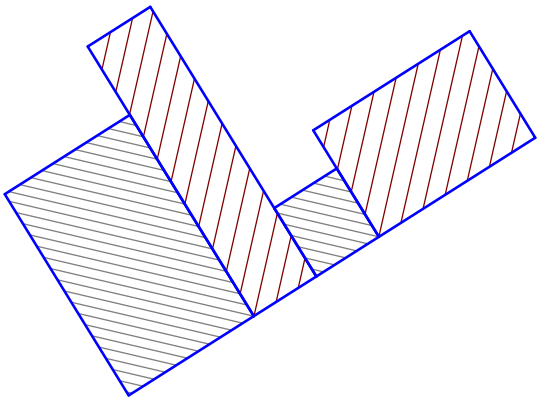


Figure 5: Objects after rotation

The dimensions are now applied. The starting point is to be the longest rectangle. The dimension should be at least 10 mm from the top edge of the highest rectangle. All the other dimensions will align themselves to this dimension. Choose **Lettering > Dimension > Length, Object**. Now, press SHIFT+ESC and the “Options” button.

This calls a dialog (Figure 6) where you can specify various options for the appearance and positioning of the dimension line, the start and end points and the positioning of the dimension figure.

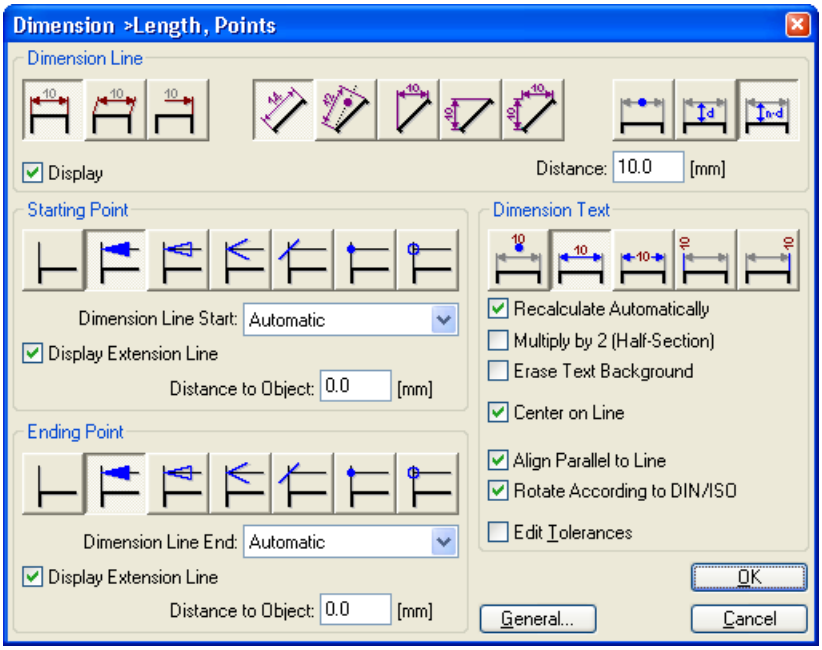


Figure 6: Dialog “Dimension > Length, Object”

Identify the segment to be measured and move the mouse towards the dimension line until it blends with the red dimension line. Confirm this by clicking the left mouse button (Figure 7).

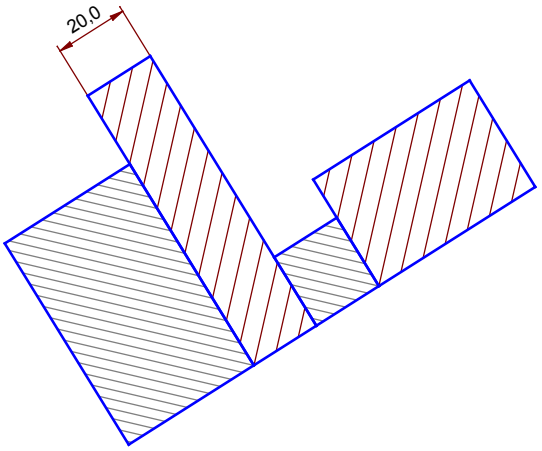


Figure 7: Measuring the first rectangle

It is necessary to alter the settings for the subsequent measurements. Either call the command **Lettering > Dimension > Length, Object** again or press SHIFT+ESC. This key combination restarts the command and calls the dialog again. To align the dimension lines correctly, it is not possible to use a fixed distance; choose the icon for user-defined spacing (see picture) and close the dialog by clicking on “OK”.

Figure 8: Icon for user-defined spacing

Identify one of the three lengths which still has to be measured. The dimension line can be moved freely. To position it accurately, choose “Corner/Endpoint” snapping mode from the panel and press the F6 key to turn snapping on. Four small lines marking the snapping radius appear round on the crosshairs. Click on the intersection of the existing dimension line and the existing extension line to position the new dimension line (Figure 9).

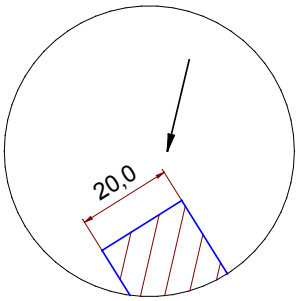


Figure 9: Zoom of the snap point

The new dimension line will snap into place correctly aligned with the other dimension line. Proceed in a similar way for the remaining dimension lines. You can now turn snapping off again, either by pressing F6 or by clicking on the button in the panel. In many cases, it can make more sense to temporarily turn snapping on or off by holding down the SHIFT key. Try both methods. Your drawing should now look like this:

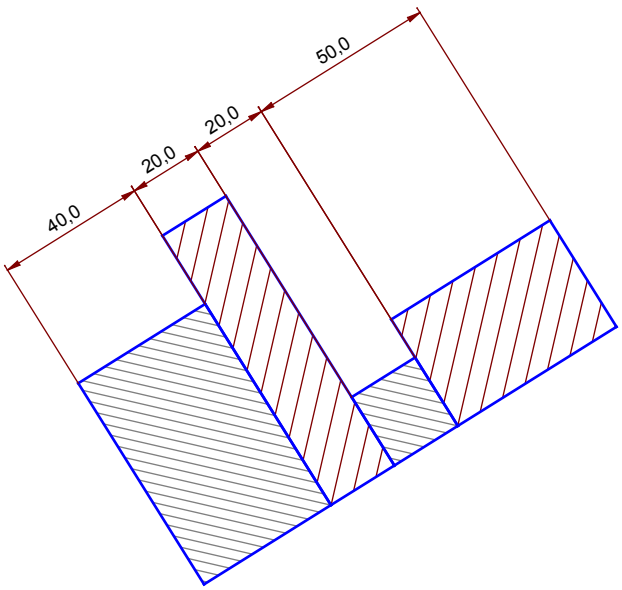


Figure 10

Next the base line will be drawn in. This is done in two steps. Choose **Draw > Line > Horizontal** and specify the starting point of the line. After clicking on the status line or pressing the F8 (ENTER) key a dialog appears into which a coordinate length or radius can be entered. By default, a "sensible" value is already entered. Enter 50 mm as the length. Choose **Shape > Move Objects > Standard** and identify the horizontal line which you have just drawn. Next it is necessary to enter a reference point. To place this exactly in the middle of the line, activate "Midpoint" snapping mode and click on the line. You have "grabbed hold" of the center of the line and can move it about. The line's mid-point should be placed on the bottom corner of the first rectangle. Activate "Corner/Endpoint" snapping mode (you may have to turn off the old snapping mode) and click on the corner (Figure 11).

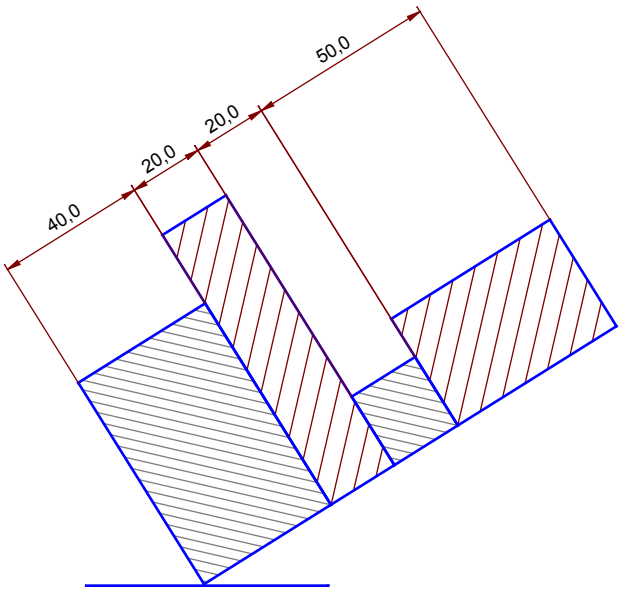


Figure 11: Baseline

Finally, the angle between the baseline and the rectangle will be added to the drawing. Choose **Lettering > Dimension > Angle, Line - Line** and identify the ends of the two lines enclosing the angle. You can now choose from multiple angles by moving the crosshair. If you want to dimension an angle larger than 180°, keep the CTRL key pressed.

In a similar way to measuring lengths, a extension line appears at a fixed radius from the center of the angle. Confirm the dimension line position 40 mm from the center of the angle (one "jump" = 10 mm) (Figure 12).

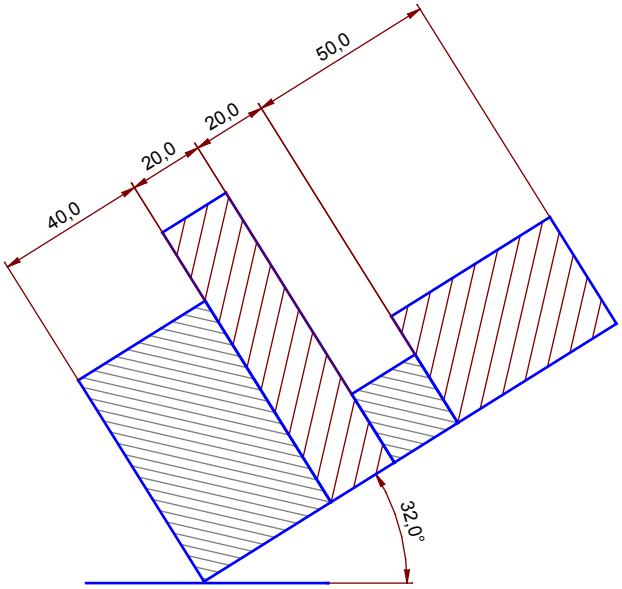
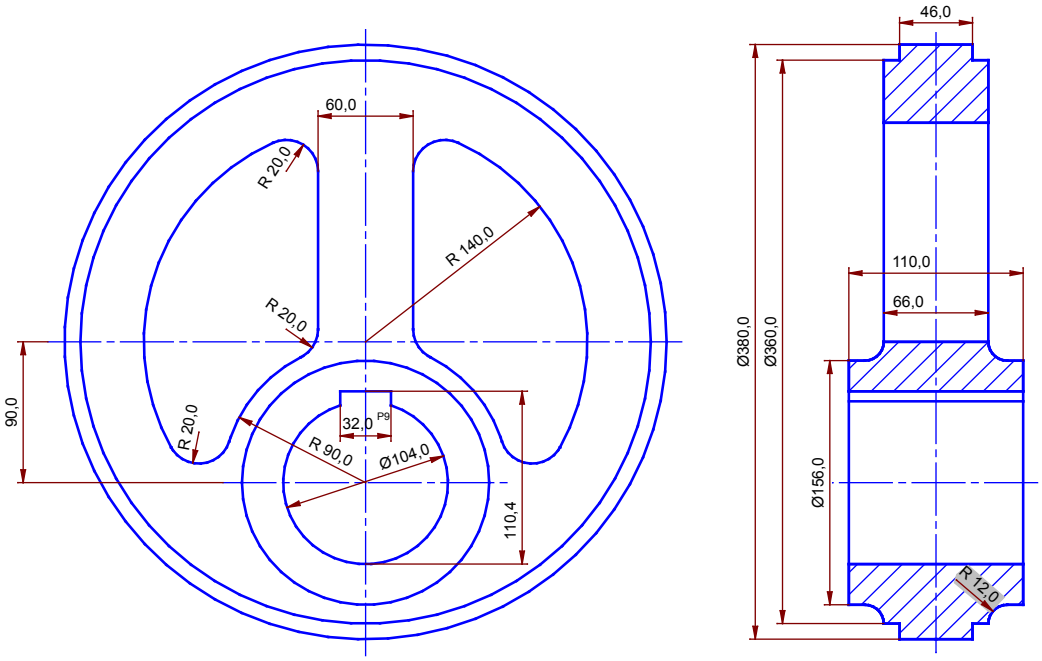


Figure 12: Angle between the baseline and the rectangle

Save your drawing using **File > Save Drawing as**.

The Eccentric



This chapter uses some commands that are only available in Malz++Kassner CAD6 Pro, CAD6 Studio, and/or CAD6 Industry!

Aim

This drawing of an eccentric is intended to show the use of the Construction Aid. Various construction procedures and measurements with differing parameters are used. You will learn how to generate and hatch surfaces and to merge objects from other drawings. To remove parts of the background, erasers are used as aids to drawing.

Settings

`Tutorial.mkd`

Remarks

Descriptions of choosing commands and working procedures are briefer than in the previous example. Entry of precise values is always done by pressing the F8 key or by clicking on the status line. The relevant values are entered in a standard dialog window if the chosen command does not have its own dialog for the entry of values.

You should already be familiar with the snapping functions and procedures for identifying objects.

Page Format

The eccentric is to be drawn on a landscape DIN A3 page in the scale 1:2. Choose **View > Pages > Edit** and click on the entry for "DIN A3" in the list of formats. Click on the icon for "Landscape" and then click on "OK".

Use the command **View > Coordinate Systems > Edit**. to set the scale. Click on the "View" button and then on Reduction Sales: 1:2 button. Close the dialog by clicking on "OK".

Because the drawing consists of a part with two views which use a common Construction Aid, it can make sense to insert an alignment mark before beginning drawing to make it easier to put the Construction Aid and the objects in the correct relationship. Alignment marks are used in the printing industry for example to line up printing films or plates precisely. In this example, the Construction Aid is on one film and the objects are on the other. If objects have to be moved during drawing, it is important to move the Construction Aid by the same amount. Alignment marks help you to do this. The coordinate system's origin, normally the bottom left of the sheet, is a good place for an alignment mark.

Choose **Construction > Construction Aid Circle > Standard**, press the F8 key and enter the coordinate `0` and `0` into the dialog. This places the center of the Construction Aid circle on the origin. Pressing F8 again and entering `2` in the dialog sets the Construction Aid circle's radius to 2 mm. Choose **Construction > Markings**, press F8 again and enter `0` and `0` to place a marking at the center of the Construction Aid circle.

You can move the Construction Aid to agree with the position of the drawing by choosing **Shape > Move Objects > Standard** and grabbing hold of the marking with "Midpoint" snapping mode active and letting go of it on the marking with "Marking" snapping mode active.

Alignment

The starting point of the drawing is to be the plan, the left part of the drawing. To guarantee horizontal alignment, place a horizontal Construction Aid line across the center of the page by choosing **Construction > Construction Aid Endless Line > Horizontal**, pressing F8 and entering the term `(_pt+_pb)/2` into the field y. This will make the program work out the y-coordinates for horizontal Construction Aid lines from the coordinates of the page's upper and lower edges. If the coordinate system origin is on the page's bottom edge, the same as the default settings in `MKCAD5.MKD`, then `_pb/2` is all that has to be entered. This entry places the horizontal Construction Aid line half way up the page.

The Plan

In addition, place a vertical Construction Aid line approximately one third of the way across the page using **Construction > Construction Aid Endless Line > Vertical** The resulting intersection marks the provisional center of the plan (Figure 1). The final alignment is done at the end.

Choose the pen “0.5 mm\Solid Line Wide” and use **Draw > Circle > Standard** to draw a circle centered on the intersection of the Construction Aid lines. Whilst doing this, briefly activate the “Construction Aid” snapping mode using the SHIFT key. After specifying the center, press F8 and specify a radius of 190 mm.

Choose **Draw > Circle > Concentric** and identify the previously drawn circle. Enter a radius of 180 mm for the new circle. Your drawing should now look similar to this:

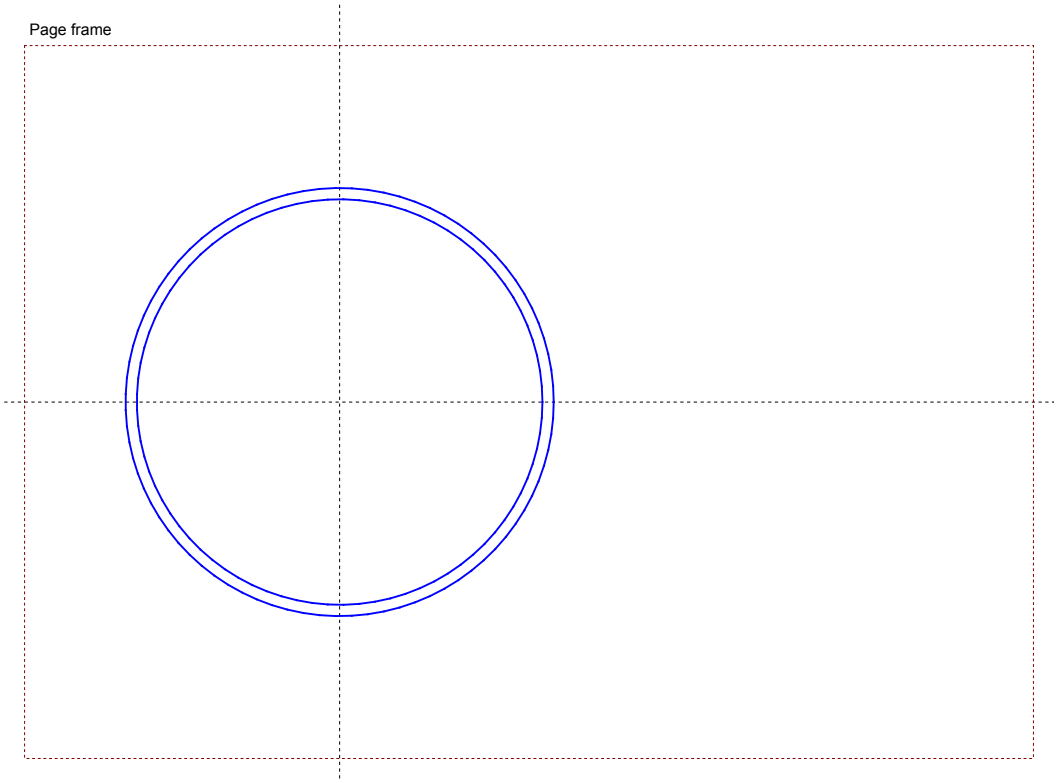
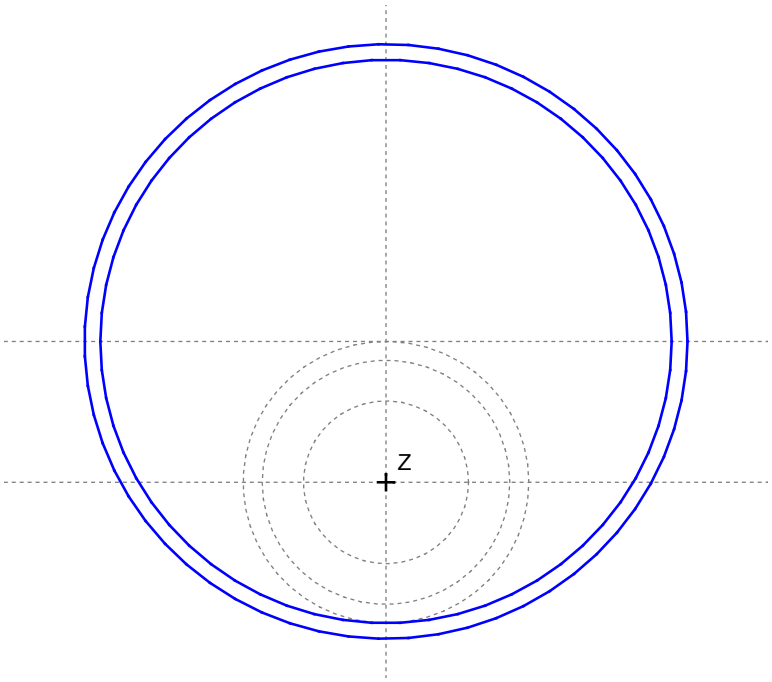


Figure 1

The Bearing

The center of the eccentric bearing is specified next. It lies 90 mm below the intersection of the Construction Aid lines. Choose **Construction > Construction Aid Endless Line > Parallel, numerical**, enter 90 mm as the distance and identify the existing horizontal line as the reference line. Place the parallel line underneath the reference line. The Intersection of this new line and the vertical line (Point Z) is the center of the bearing. (Figure 2).



Using this new intersection as the center, use **Construction > Construction Aid Circle > Standard** to draw a Construction Aid circle with a radius of 52mm. Use **Construction > Construction Aid Circle > Concentric** to draw 2 further circles with radius of 78 mm and 90 mm (Figure 2).

The two circles can be drawn in the same command sequence. Instead of finishing the command after entering the first radius, press F8 again, enter the second radius and then finish the command (click right mouse button, and if the Workflow Manager is active and appears, choose **Finish Command**).

The Spring Nut

These Construction Aid lines are used to determine the position of the spring nut. Use **Construction > Construction Aid Endless Line > Parallel, numerical** to draw two parallel vertical lines 16 mm on either side of the existing parallel line. This can be done in one step by first identifying the vertical line then clicking with the left mouse button to the left of the reference line and clicking with the left mouse button to the right of the reference line, then finishing the command sequence (click right mouse button, and if the Workflow Manager is active and appears, choose **Finish Command**).

To determine the depth of the nut, a horizontal Construction Aid line is placed through the lower quadrant of the innermost Construction Aid circle. Choose “Quadrant” snapping mode and draw a horizontal line through the lower quadrant using **Construction > Construction Aid Endless Line > Horizontal** with the SHIFT key held down. The spring nut is 110.4 mm from this line. **Construction > Construction Aid Endless Line > Parallel, numerical** is used to specify the distance exactly and draw the Construction Aid line.

The outline of the bearing is found from the intersections. Turn on the “Construction Aid” snapping mode and use **Draw > Line > Polyline** to draw the bearing’s sprung nut by using the intersections (A, B, C and D in Figure 3) as the corners of the polyline. End the command sequence by clicking the right mouse button after entering the fourth corner point.

If you misplace one of the corner points, then you can take it back by pressing the ESC key before ending the command.

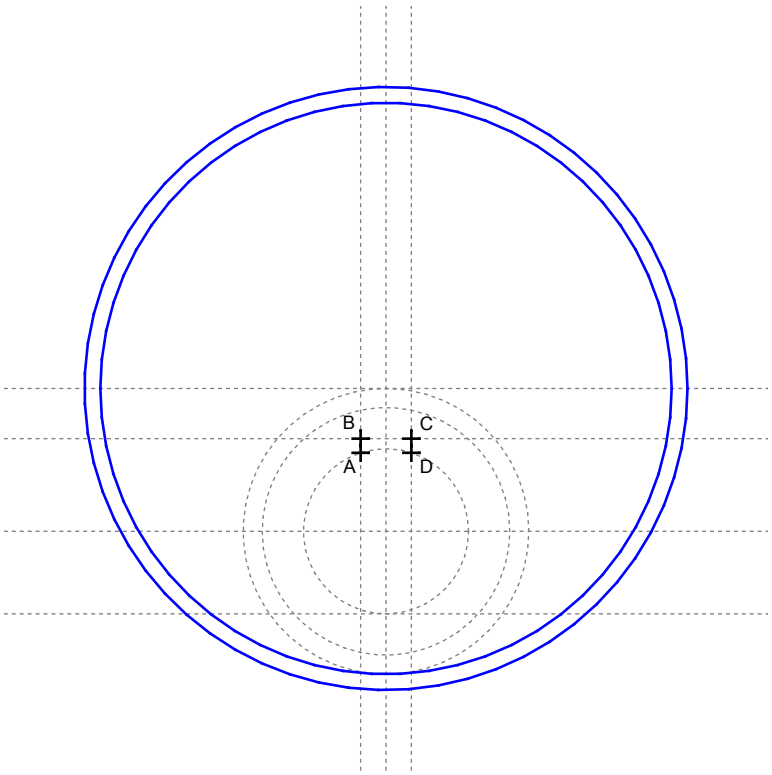


Figure 3

Complete the bearing by drawing the remaining arc using **Draw > Arc > On Circle** Identify the Construction Aid circle on which the arc is to be drawn and then choose as start and end points the ends of the polyline. You can have the “Corner/Endpoint”, or “Construction Aid” snapping modes, or both, active.

If the arc runs in the wrong direction, you can easily change it during drawing using the relevant button from the panel. If you have finished drawing the arc in the wrong direction, change it by using the command **Trim > Transform Object to > Inverted Object**

Use **Draw > Circle > On Circle** to complete the middle concentric circle (see Figure 4).

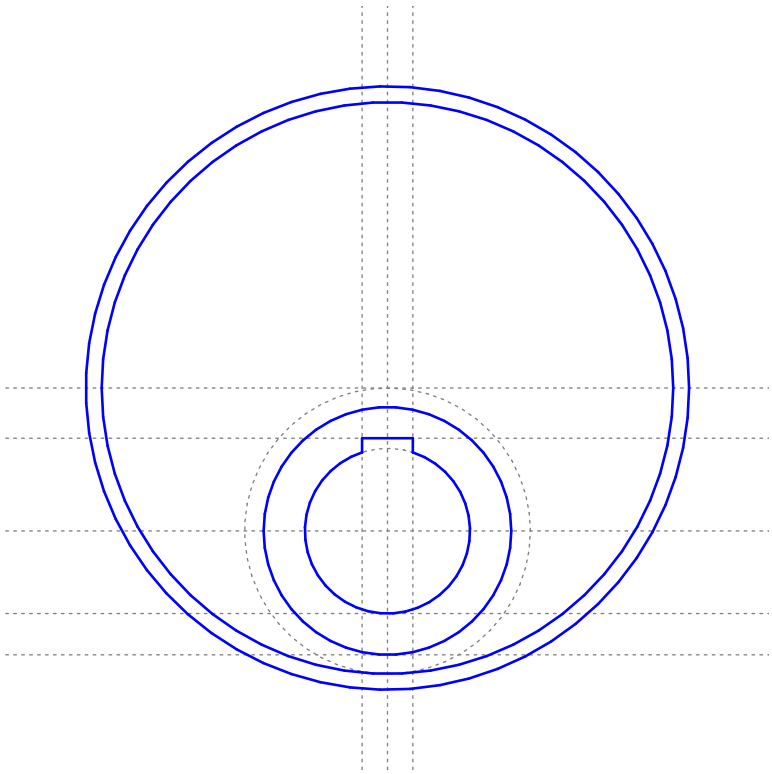


Figure 4

The Blank Spaces

The blank spaces in the eccentric are still missing. Only one has to be drawn: the other can be created using mirroring with the duplicate function turned on. The outline of each cut-out consists of the five circle parts (3×1, 2, and 4) as well as a line (3) (Figure 5). Of the circle parts 2 and 4 as well as the line 3, only the position of the Construction Aid on which they sit is known. The start and end points are not known.

The three circle parts 1 have a known radius, but their center points are not known. Their position must therefore be worked out from further information. This extra condition is that each of the three circle parts ends tangential to two further objects.

First it is necessary to draw the missing Construction Aid. Construction Aid circle 1 already exists. Use **Construction > Construction Aid Circle > Concentric** to draw a Construction Aid circle with a radius of 140 mm to use for circle part 2. In addition, use **Construction > Construction Aid Endless Line > Parallel, numerical** to draw a line 30 mm distant from the vertical Construction Aid line. The line (3) from the cut-out will be drawn there later.

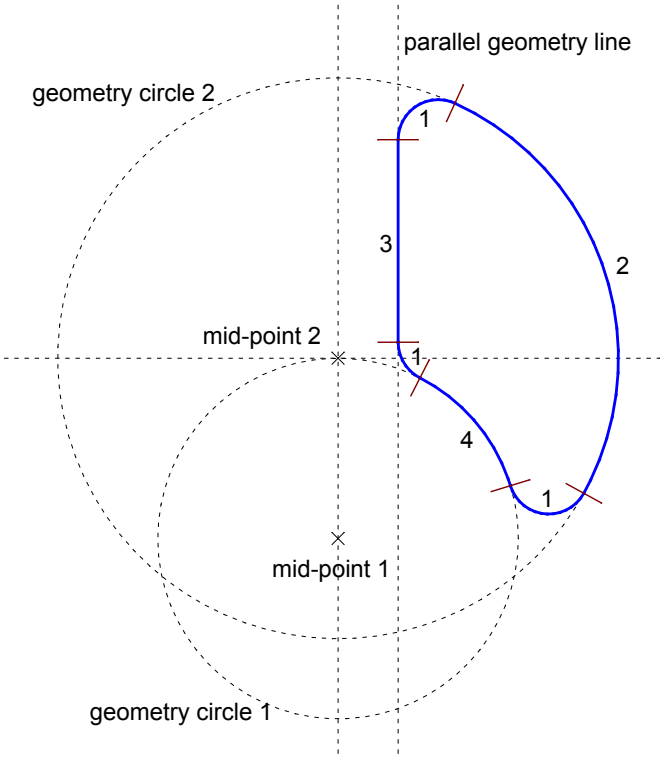


Figure 5

Begin to construct circle part 1 with **Draw > Arc > Radius - Object - Object** . Enter the radius 20 mm in the dialog field. Now identify both the Construction Aid elements to which the arc should be tangent. For the upmost arc, these are the last-drawn vertical Construction Aid line and Construction Aid circle two. The program will offer a choice of the arcs which it can draw. Click on the “Next” button until the variant which you want appears and then click on “OK”. Repeat the process to draw the other two arcs 1 (Figure 6).

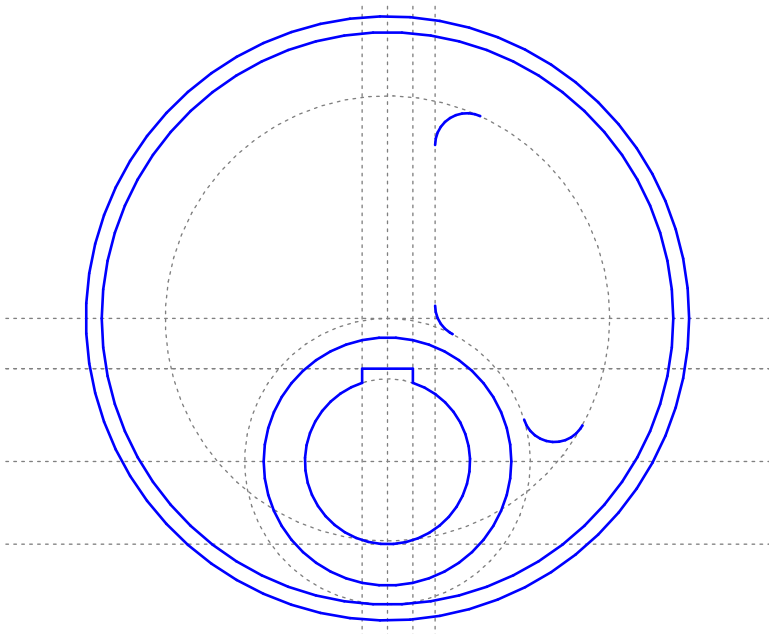


Figure 6

Because the tangential points are now determined, the elements two, three and four can be drawn on top of the existing Construction Aid. Choose **Draw > Line > Standard** and draw the missing straight line by clicking on the two arcs’ endpoints with “Corner/ Endpoint” snapping mode active.

Choose **Draw > Arc > On Circle** and draw the missing arc by identifying the relevant Construction Aid element and clicking on the two arcs' endpoints with "Corner/Endpoint" snapping mode active (Figure 7).

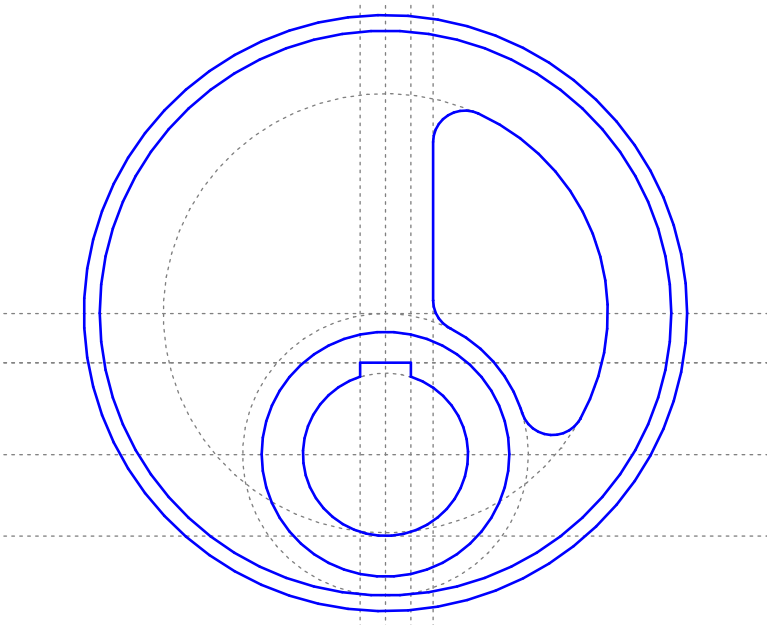


Figure 7

The outline for the right cut-out is now complete. Choose **Shape > Reflect Objects > Line**. To be able to mirror and duplicate the cut-out in one move, press and hold down the CTRL key and identify each of the elements in the cut-out's outline. Holding down the CTRL key makes it possible to identify more than one object for manipulation at the same time. Release the CTRL key after identifying all the elements in the outline. Next, it is necessary to identify the axis of reflection. Press the CTRL key again to activate the duplicate function. You will know that it is active because a "+" appears on the crosshair. Identify the vertical Construction Aid line running through the center of the circle as the rotation axis by clicking on it. The reflection should now appear symmetric to the original (Figure 8).

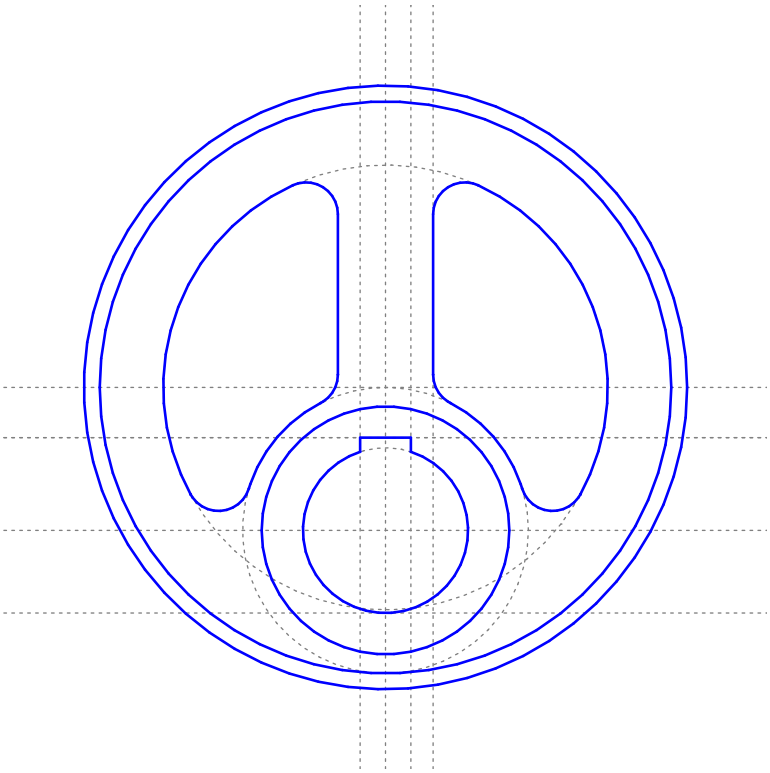


Figure 8

Apart from the mid lines, the section indicator and the dimensions, the plan is now complete. Before they are added to the drawing, the section drawing should be drawn.

The Section View

To make drawing the section as easy as possible, the most important points from the eccentric are to the side view with the help of the Construction Aid. With the “Quadrant” snapping mode active, use **Construction > Construction Aid Endless Line > Horizontal** to insert horizontal Construction Aid lines at the upper and lower quadrants of the three outermost concentric circles and the two inner bearing circles. This produces a total of nine new horizontal Construction Aid lines. The innermost bearing circle already has a Construction Aid line in its lower quadrant which does not have to be drawn again (Figure 9).

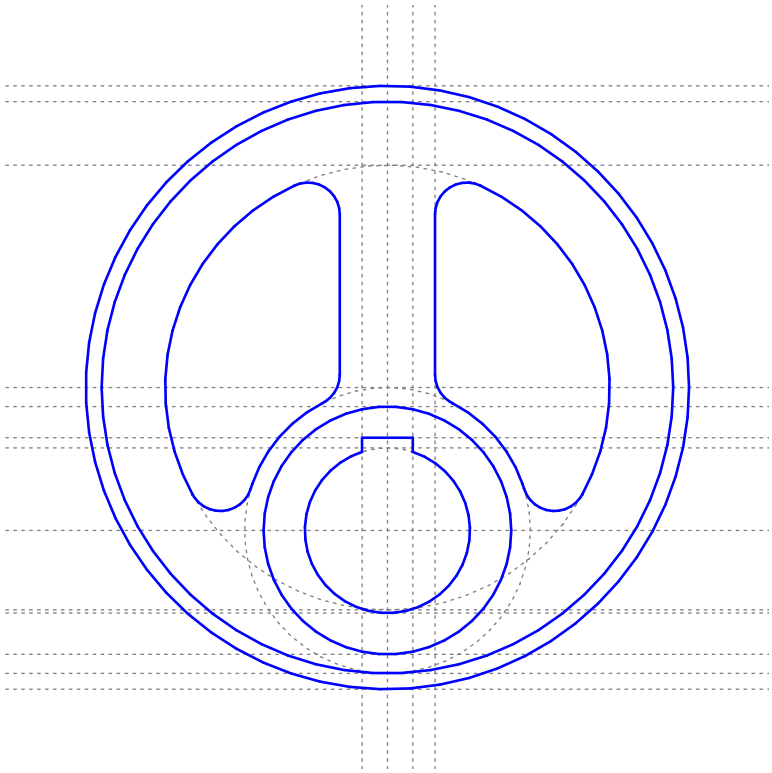


Figure 9

Use **Construction > Construction Aid Endless Line > Parallel, numerical** to place a vertical Construction Aid line 360 mm to the right of the eccentric’s central vertical Construction Aid line. This line is the central vertical of the sectional drawing. If it proves necessary to move the view to get space, we remind you of the tip concerning moving the Construction Aid at the start of this chapter.

The page should now look like this:

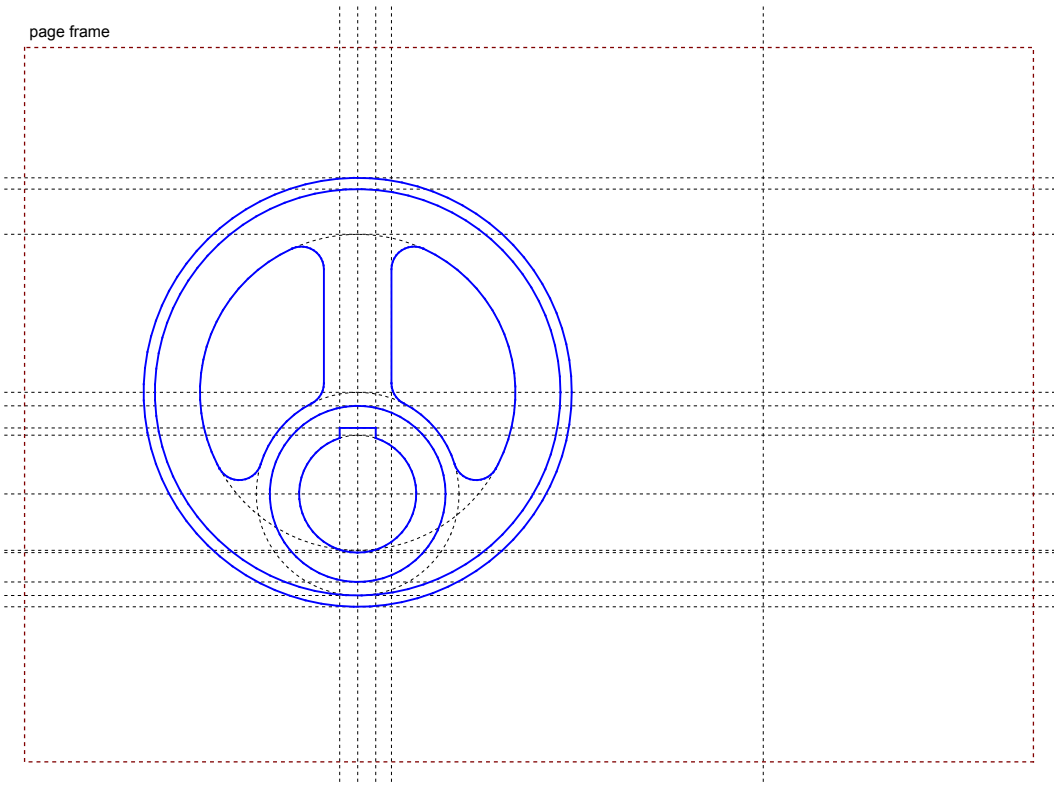


Figure 10

Because the sectional view is completely symmetrical, it is enough to draw half of it and then mirror all the objects about the center line with the duplicate function turned on. You have already used this function whilst drawing the cut-outs in the plan view.

Choose **Construction > Construction Aid Endless Line > Parallel, numerical** and place three vertical lines 23 mm, 33 mm and 55 mm to the right of the axis of rotation. The easiest way of doing this is to press SHIFT+ESC (N) to restart the command and enter a new value.

All the intersections needed to draw the sectional view are now in place. Nest, the rounded shapes for the bearing are drawn. They both have a radius of 12 mm and are tangent to two Construction Aid lines. Choose **Draw > Arc > Radius - Object - Object**, enter the radius 12 mm and identify the tangential Construction Aid lines. Choose the desired variant.

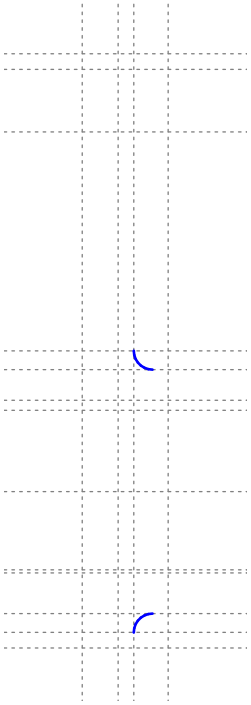


Bild 11

Now the remaining intersections are joined using a polyline drawn using **Draw > Line > Polyline** and the “Construction Aid” snapping mode turned on (Figure 12).

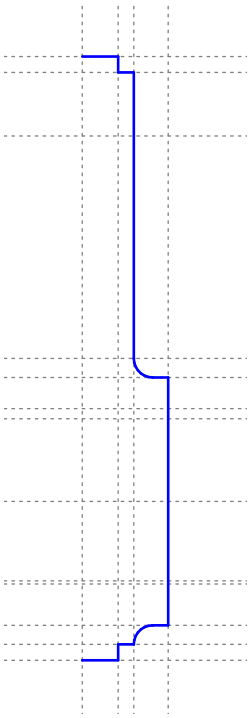


Figure 12

Choose **Shape > Reflect Objects > Line** and, holding down the SHIFT key, drag out a rectangle around all parts of the sectional drawing. Release the key. This has identified the objects for the following operation. Press and hold down the CTRL key to activate the duplicate function and identify the left of the four vertical Construction Aid lines as the reference line (axis of rotation).

To complete the outline, draw the five horizontal lines using **Draw > Line > Standard** with both “Intersection” and “Construction Aid” snapping modes active. This is needed to ensure that the lines snap correctly to the intersection points on the reflected side of the drawing as the objects intersect with both horizontal and vertical Construction Aid lines (Figure 13).

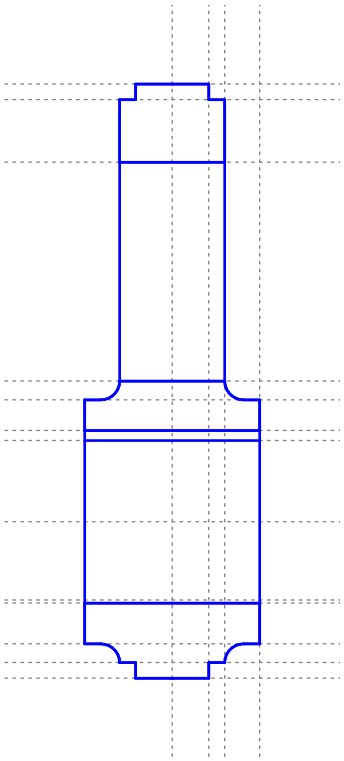


Figure 13

Surfaces and Hatchings

The Construction Aid is not needed during the next step. It is best if you turn it off with the button in the panel or with **Construction > Display Construction Aid**.

Three areas in the sectional view need to be hatched. It is necessary to temporarily generate a surface from the objects surrounding the areas to be hatched. To generate a surface, all the objects surrounding the surface have to be identified. Make sure that the button for “Inside” area mode is pushed in. Change to the pen “0.25 mm\Solid Line Narrow” and choose **Draw > Hatching > Generated Surface** and, holding down the SHIFT key, drag a box around the surface which is to be generated. The future surface must lie completely within the box, or all the line parts may not be identified. Specify a reference point inside the area of the new surface. Then terminate the point entry (click right mouse button, and if the Workflow Manager is active and appears, choose **Finish Command**).

Carry out a similar procedure for the other two surfaces (Figure 14).

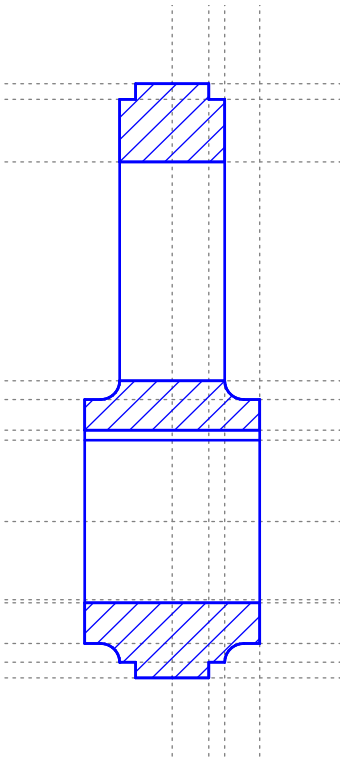


Figure 14

Additions

Now, the symmetry lines are added. Turn the Construction Aid on, choose the pen “0.25 mm\\Dash-Dot Line Narrow” and draw the symmetry lines on the Construction Aid lines in the sectional view using **Draw > Line > On Line**. The lines should extend a little way beyond the eccentric’s edge.

The center lines on the plan view are drawn simply by choosing **Draw > Line > Center Cross** and snapping the outermost circle's center-point. The horizontal center line of the bearing is also drawn using **Draw > Line > On Line**. The Construction Aid can now be finally turned off (Figure 15).

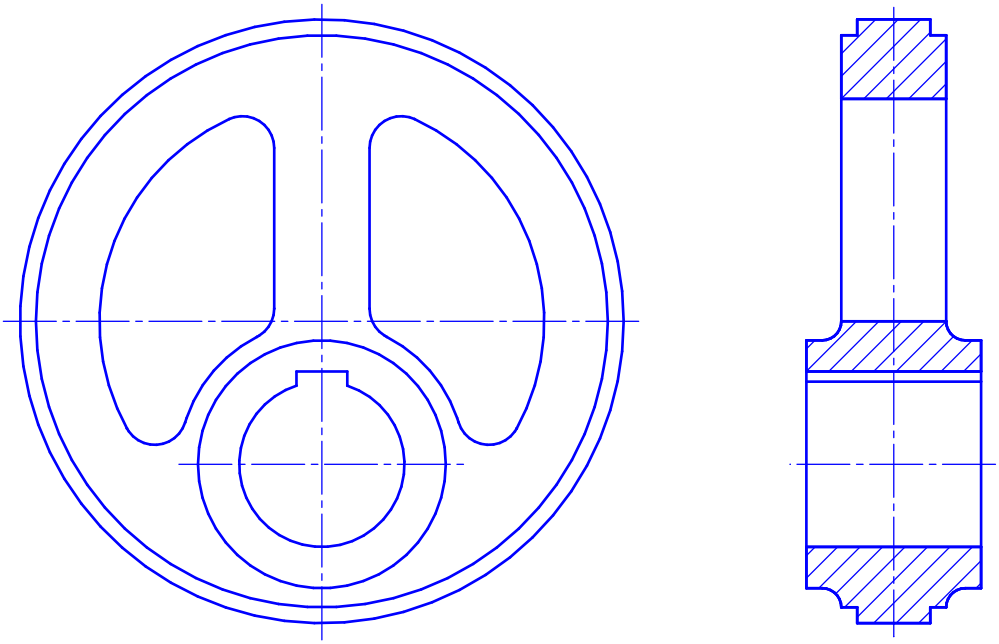


Figure 15

Adding Dimensions to the Sectional View

Choose the pen "0.25 mm\Solid Line Narrow". With the exception of the radius, all the dimensions are applied to the sectional view using **Lettering > Dimension > Length, Point** The preset parameters can be used for the dimensions outside the objects. To work out the start and end points of the lengths, activate the "Corner/ Endpoint" snapping mode. For the dimensions inside the body, restart the command using SHIFT+ESC and clear the "Center Dimension" check box. This makes it possible to position the dimension to the left of the central vertical line. For the 66 mm width dimension turn off the check box for extension lines as well and snap to the lower corners of the topmost surface. Because there are no extension lines, the dimension can be moved downwards as far as wished.

Next, choose **Lettering > Dimension > Radius Object**, identify the radius and place the dimension in the hatching.

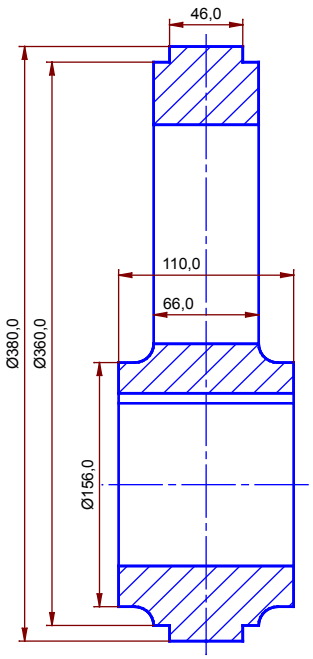


Figure 16

The Eraser

To clear an area for the radius' dimension in the hatching, use the eraser. Every object which is drawn has a position relative to the objects drawn before and after it. You can think of the objects as playing cards, which, when placed on the pack, partly or completely cover the underlying cards.

Use the command **Draw > Polygon > Arbitrary** to place a surface over the dimension which will "rub out" the background and only allow those objects to appear which are "above" that surface (i.e. drawn later). Then, choose the command **Shape > Edit Properties** identify the previously drawn surface and set its filling mode to "Eraser" (the fourth of the five available filling modes).

Objects can be brought to the front or sent to the back. This is necessary in this case because the eraser - to allow calculation of the dimension - was the last-drawn object. Choose **Shape > Change Order > To Front** and click in the center of the erasing surface, under which the dimension lies. (It is also possible to use **Shape > Change Order > To Back** and move first of all the erasing surface and then the hatching to the back.) Bear in mind that the erasing surface must be identified at its edge. The erasing surface should now block out part of the hatching and the dimension should be clearly visible (Figure 17).

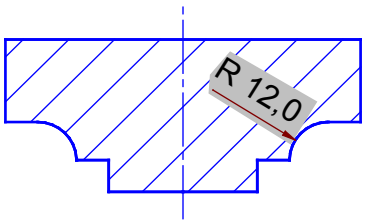


Figure 17

Add the 'Ø' symbol to the vertical dimensions by choosing **Shape > Edit Text** identifying each dimension in turn and entering the 'Ø' symbol in the Pre-Text field (do this by clicking on the relevant button) and clicking on "OK".

Adding Measurements to the Plan View

Now, measurements are added to the plan view. Firstly, apply measurements to the 20 mm radii in the left-hand cut-out. Choose **Lettering > Dimension > Radius, Object** and deactivate the "Center Dimension" check box in the parameter dialog (accessible by pressing SHIFT+ESC). This makes it possible to place the dimension at a suitable point on the dimension line. Identify a radius, choose the dimension line direction between the center point and the radius and place the end of the line at the center by clicking on the circle's edge with "Midpoint" snapping mode turned on. Pull the dimension to the correct position.

Proceed in the same way with the cut-out's 90 mm and 140 mm radii. The "Center Dimension" check box should also be turned off when applying dimensioning to the inside diameter of the bearing. This is necessary because a separate set of parameters is stored for each dimensioning variant. For the inner radius choose **Lettering > Dimension > Diameter, Object**, to put the 'Ø' character in the Pre-Text automatically (Figure 18).

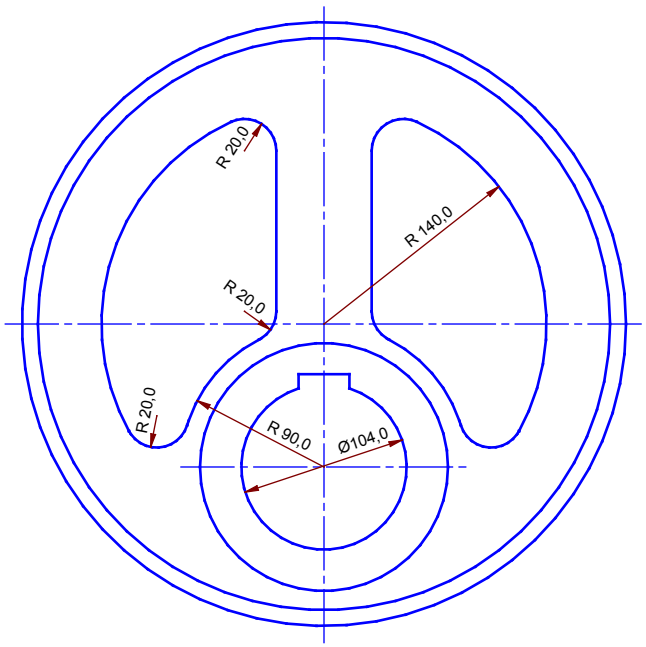


Figure 18

Next, the length measurements are applied. Because none of the measurements is concerned with a single object, the command **Lettering > Dimension > Length, Point** must be used. The corners at which the 32 mm and 60 mm extension lines begin are used as snapping points. The dimension also has to be positioned by hand in this case. Do not forget to turn the extension lines on again. To measure the 32 mm distance, activate the Tolerances check box in the dimension parameters dialog so that you can enter P9 as the upper tolerance in the Tolerance dialog which appears later.

To apply a dimension of 110.4 mm to the sprung nut in the drawing, use the top intersection of the nut with the central vertical line and the and the quadrant point of the circle (or the intersection of the circle and the central vertical line) as snapping points.

Finally, the distance from the midpoint to the center of the eccentric (90 mm) is added. So as not to cover the dash-dot symmetry line, the extension lines can only start at the ends of the two dash-dot lines. It is necessary to keep the dimension lines vertical. Do this by activating the “Center Dimension text” and “Adapt Dimension text” check boxes in the Dimension dialog, activate “Corner/Endpoint” snapping mode and click on both ends of the dash-dot line. You can now set the dimension line’s orientation. Press the F8 key and enter 90°. Now you have only to position the dimension line in the usual manner (Figure 19).

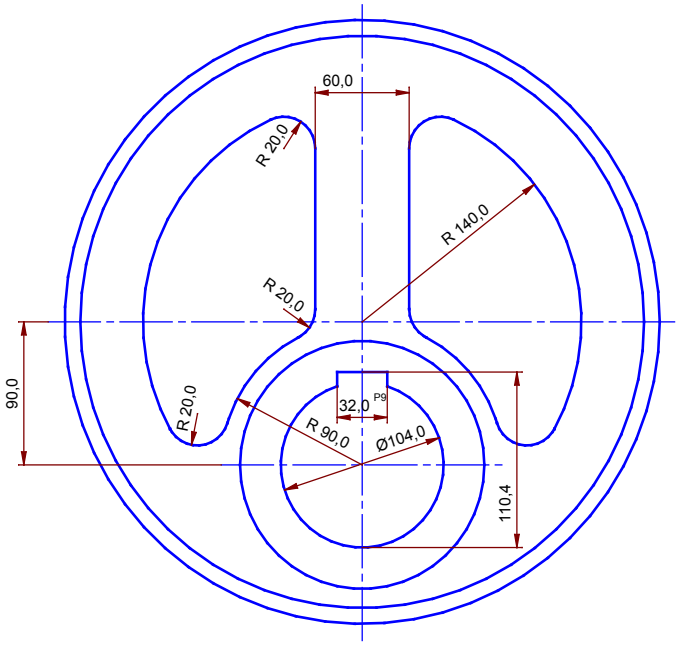


Figure 19

Drawing Frame and Text Box

Now a standard template is to be added. Select all the objects in the drawing with **Extra > Permanent Selection > Set** and pressing F10. This makes it possible, after inserting the template, to move the selected objects but not the newly added ones.

Now open the library containing those templates (`001_Title_Blocks_DIN_6771.mkl`) using the **Library > Library Management** command. Then use the **Library > Block > Insert** command to choose the `DIN A3 Landscape` block in the `Frames` folder and place it in the drawing. Once finished, select the `Type 4` block from the `Title Blocks` folder and place it in the lower right corner of the frame.

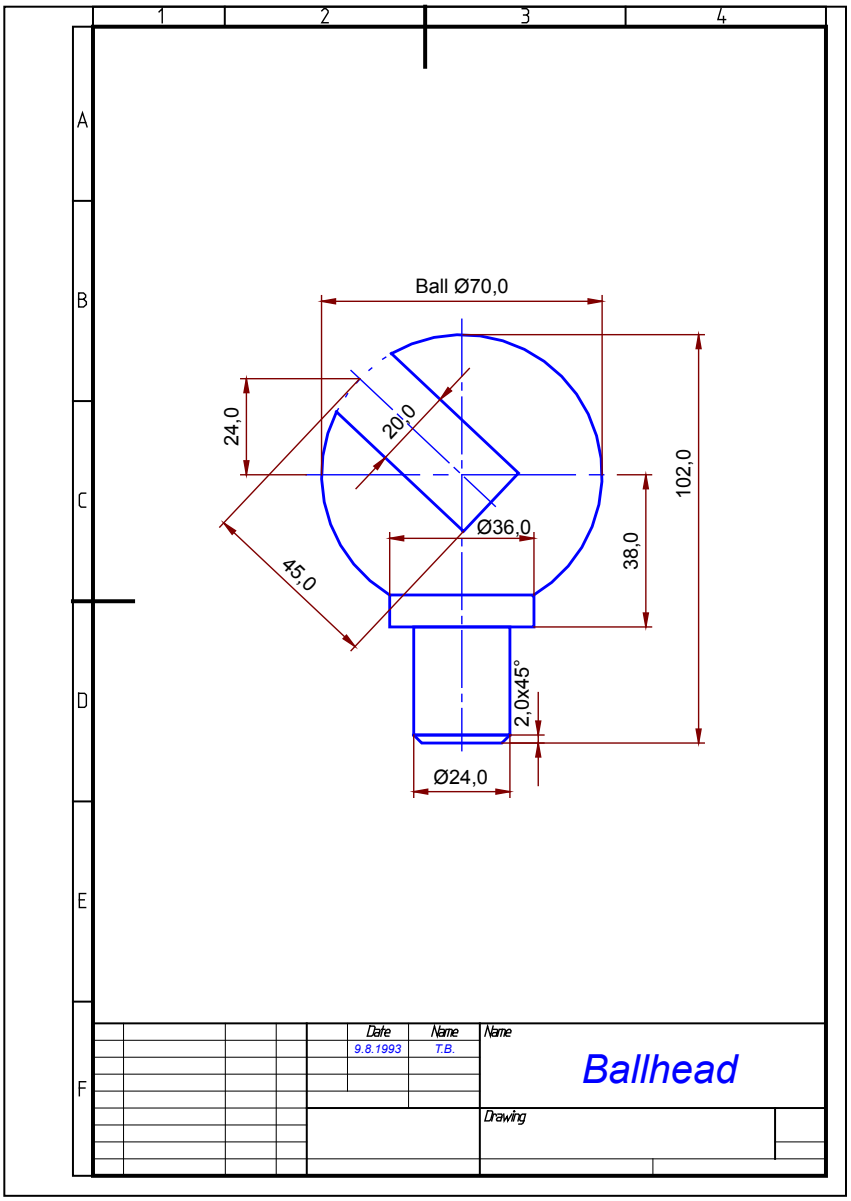
Before the next step, you should make sure that you placed one of the pass marks referred to earlier otherwise it will be much harder to align the Construction Aid after the centering. Center all the selected objects in the free drawing area using **Shape > Align Objects > Center, Frame Both**. To identify all the selected objects (but not the merged objects) press the F12 key and choose as the area in which everything should be centered the inner upper left corner of the drawing frame and the upper right corner of the text box. All the selected objects can now be deselected by choosing **Extra > Permanent Selection > Clear** and pressing F10.

Finally you can use the command **Lettering > Text > Standard** to generate the necessary text to go in the text box. Save the drawing using **File > Save Drawing as**.

Alternative Construction Methods

The sprung nut and the cut-outs can also be constructed by using standard objects and trimming them. Try to draw the eccentric without referring to the tutorial. Use your own ideas and in case of difficulty, refer to the electronic reference.

The Ballhead



This chapter uses some commands that are only available in Malz++Kassner CAD6 Pro, CAD6 Studio, and/or CAD6 Industry!

Aim

This example should make both principle methods of drawing using the Construction Aid and trimming functions. Note that in practice using just one method of construction is very rare.

Settings

Tutorial.mkd

The Ballhead (Construction Aid)

Use **Construction > Construction Aid Endless Line > Horizontal** and **Construction > Construction Aid Endless Line > Vertical** to draw to lines (A and B) whose intersection places the center of the ball head as close as possible to the center of the page.

Next, draw a Construction Aid circle with a radius of 35 mm using the intersection as the center (**Construction > Construction Aid Circle > Standard**). Use **Construction > Construction Aid Endless Line > Horizontal**) to draw a horizontal Construction Aid line through the upper quadrant point of the Construction Aid circle.

Use **Construction > Construction Aid Endless Line > Parallel, numerical**) to draw horizontal Construction Aid lines 102 mm below the Construction Aid line through the upper quadrant point and 38 mm below the horizontal center Construction Aid line. The Construction Aid should now look like Figure 1.

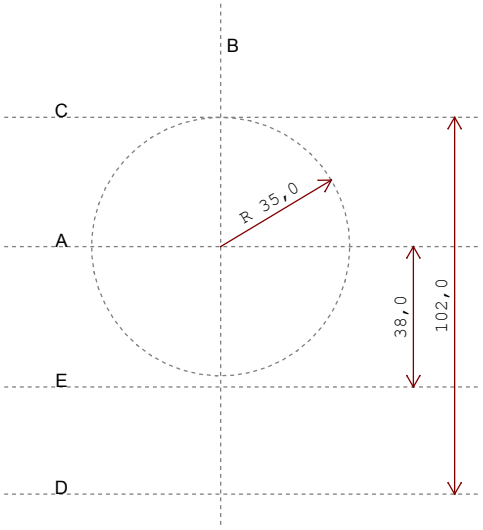


Figure 1

Next, construct parallels 12mm and 18 mm to the right and left of the central vertical Construction Aid line using **Construction > Construction Aid Endless Line > Parallel, numerical** (Figure 2).

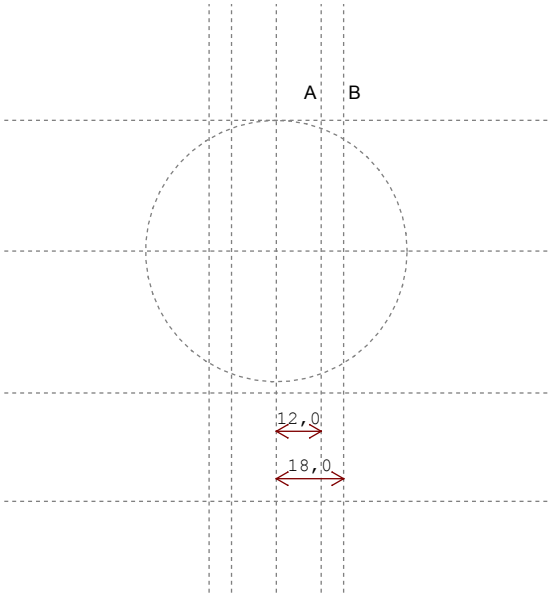


Figure 2

Draw a parallel Construction Aid line 2mm above the bottom horizontal Construction Aid line. Use the key combination SHIFT+ESC to restart the command. This new Construction Aid line will be used later when drawing the chamfered section.

Use **Construction > Construction Aid Endless Line > Angle to Line** to draw a line at 45° on the right side and -45° on the left side to indicate the position of the chamfer. Use the intersection of the 12 mm vertical Construction Aid line with the new horizontal Construction Aid line (S) as the snapping point. Repeat the process with the angle of 45° on the opposite side (Figure 3).

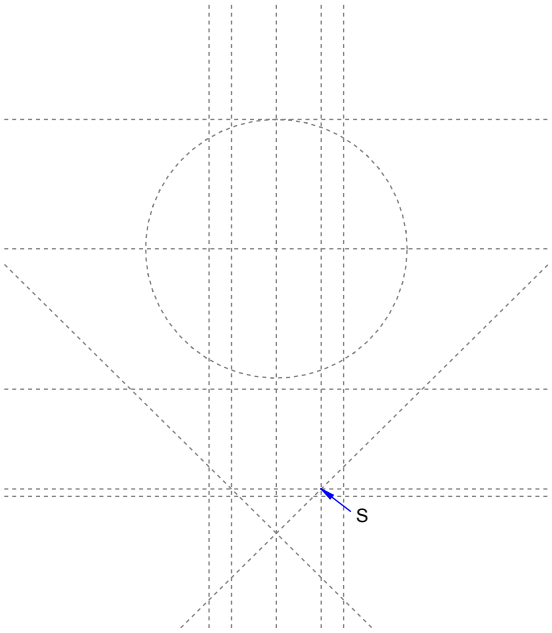


Figure 3

Draw a horizontal Construction Aid line 24 mm above the horizontal symmetry line using **Construction > Construction Aid Endless Line > Parallel, numerical**.

The left hand intersection of this new Construction Aid line with the Construction Aid circle is one end of the center line of the ball head's blank space (Figure 4, K). The other end of this center line lies at the center of the circle (M). Use **Construction > Construction Aid Endless Line > Standard**.

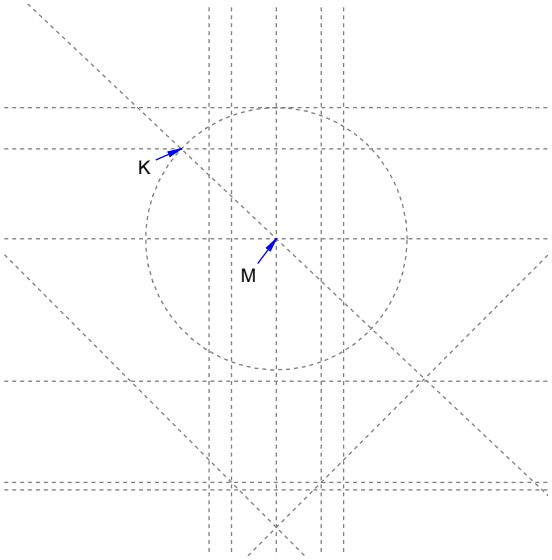


Figure 4

Use **Construction > Construction Aid Endless Line > Parallel, numerical** to draw lines 10 mm above and below this line (Figure 5).

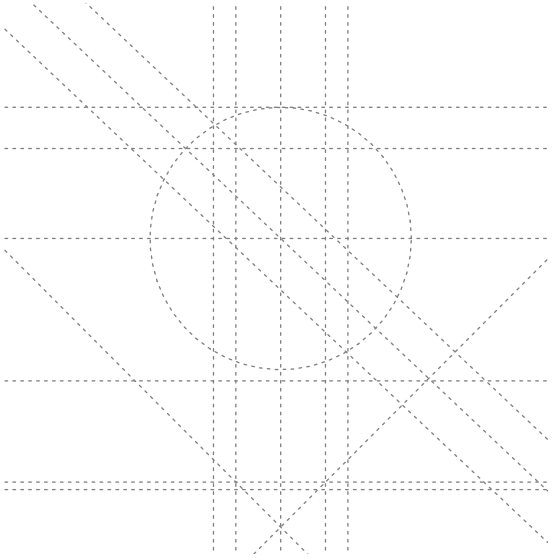


Figure 5

To determine the depth of the blank space, use **Construction > Construction Aid Endless Line > Angle to line** to draw a line at 90° to the center line M which passes through the intersection of the center line M with the Construction Aid circle. Use **Construction > Construction Aid Endless Line > Parallel, numerical**) to draw parallel B 45 mm from the new Construction Aid line (Figure 6).

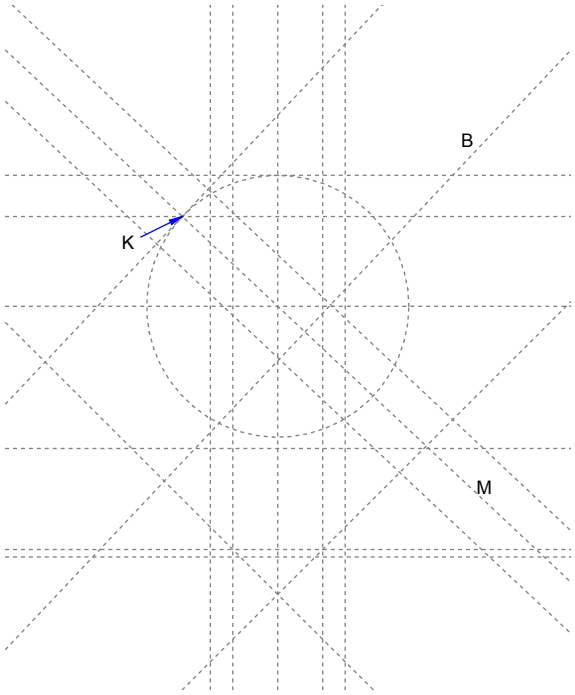


Figure 6

From the Construction Aid to the Objects

So far, you have drawn a rather complex Construction Aid. It contains all the intersections and elements needed to draw the objects. Now you can begin to draw the objects themselves. Make use not only of the “Construction Aid” snapping mode, but also of the zoom functions, as some of the intersections are close together.

Choose the pen “0.5 mm\Solid Line Wide” and begin to draw the two circle parts using the command **Draw > Arc > On Circle** (Figure 7).

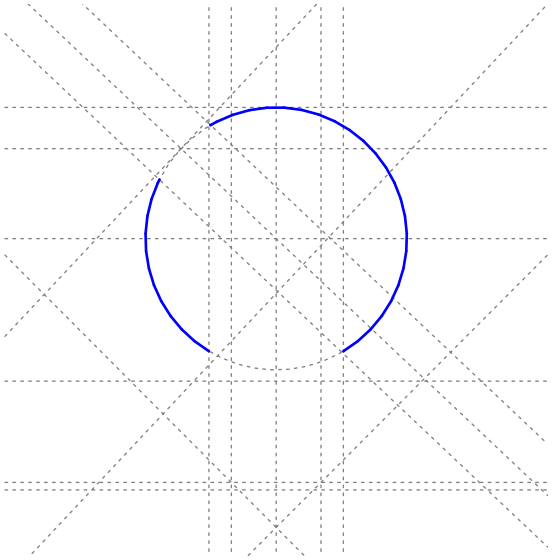


Figure 7

The next step is to draw the two rectangles underneath the ball head using the command **Draw > Polygon > Rectangle** Check the position of the rectangles against the original drawing (Figure 8).

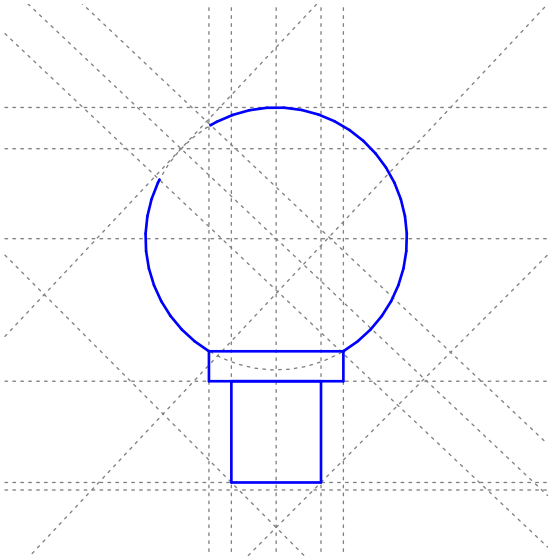


Figure 8

Complete the object by drawing the missing line sections (the blank space and the chamfered section) using **Draw > Line > Polyline**. Note that the bottom corner point of the blank space does not lie on the central vertical Construction Aid line but is a product of the two diagonal Construction Aid lines. (Zoom in on this area before locating the intersection). See Figure 9.

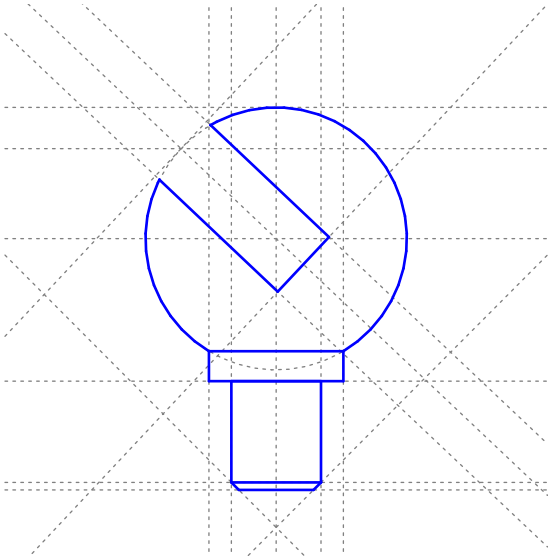


Figure 9

Next, the central lines are drawn. Select the pen “0.25 mm\\Dash-Dot Line Narrow” and draw the missing circle part (in the blank space) using **Draw > Arc > On Circle**.

Use **Draw > Line > Center Cross** to add the ball head’s horizontal and vertical center lines at the same time. To extend the vertical line below the bottom of the object, choose **Trim > Trim Object > Length / Radius to Point**, identify the lower part of the line and click on the point to which the line is to be extended. Use **Draw > Line > On Straight Line** to draw the blank space’s center line.

The Construction Aid can now be turned off using the button in the panel or by pressing F9 (Figure 10).

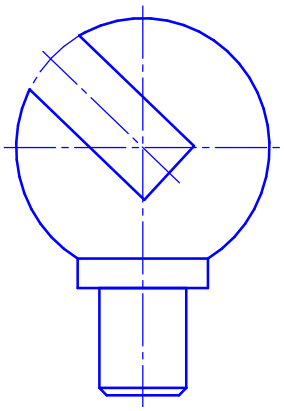


Figure 10

The 24 mm and 36 mm dimensions can be added using **Dimension > Dimension > Length, Object** and simple identification. Choose whatever distance you wish from the object in the parameters dialog (accessible by pressing SHIFT+ESC). To place the 36 mm dimension turn the “center dimension” check box off.

The 20 mm width of the blank space is measured using the command **Dimension > Dimension > Distance, Object - Object**. To make the dimension arrows point inwards choose the Orientation: Rotated setting in the parameters dialog.

All of the other dimensions are added using **Dimension > Dimension > Distance, Object - Point**. Choose the dimension line’s orientation from the icons in the dialog box.

Your drawing should now look like figure 11.

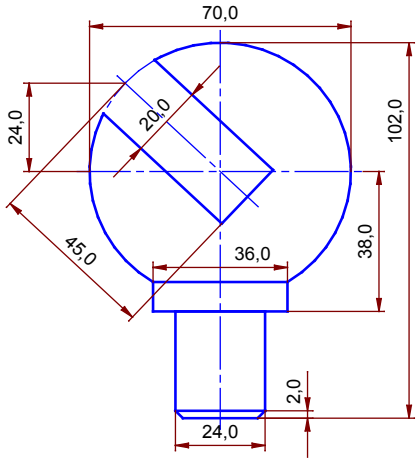
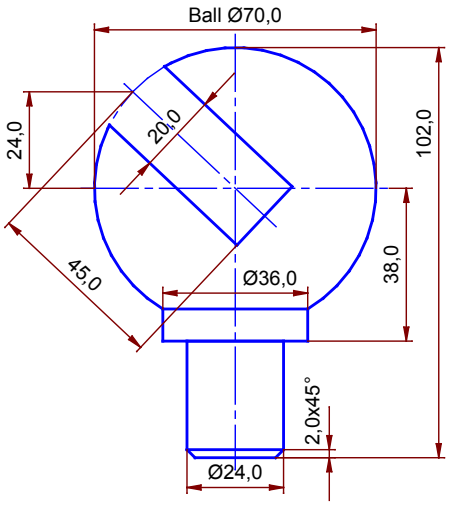


Figure 11

Finally, add the missing text and symbols using **Shape > Edit text**. If the chamfer dimension text (2.0×45°) lies within the dimension lines, you can correct this using the command **Dimension > Edit Dimension > Position** by clicking first on the dimension text and then on its new position (Figure 12).



In conclusion, as in the previous example of the eccentric cam a frame can be placed around the drawing, a text box added and the ballhead centered in the frame.

The Ballhead (Trimming)

When working with trimming functions, in general complete objects are drawn and then cut to size using the trimming functions. Even if you prefer to work using Construction Aid functions, it is a good idea to work through the following example as it explains several important principles for working with trimming functions.

Choose the pen “0.5 mm\Solid Line Wide” and draw a rectangle 38 mm high and 36 mm wide approximately in the center of the page using the command **Draw > Polygon > Rectangle**. Now construct a circle centered on the midpoint of the top side of the rectangle with a radius of 35 mm using the command **Draw > Circle > Standard** and with “Midpoint” snapping mode active (Figure 13).

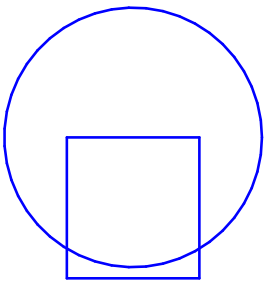


Figure 13

After choosing **Trim > Trim Object > Cut Out** click inside the part of the circle which overlaps the rectangle and then, with “Intersection” snapping mode on, click on the two points where the circle and rectangle intersect. Your drawing should now look like this:

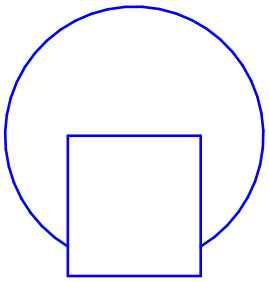


Figure 14

Now use **Trim > Trim Object > Resolve Completely** to split the rectangle into individual lines. Press the DEL key (this activates the **Shape > Delete objects** command) and click on the top edge of the rectangle. If the complete rectangle disappears, then the rectangle was not split up properly; use the UNDO command (**Edit > Undo**) to restore the rectangle (Figure 15).

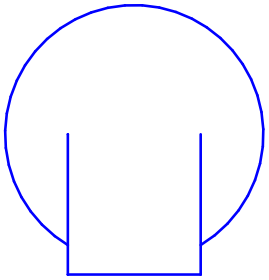


Figure 15

Join the ends of the arc by using **Draw > Line > Standard** with “Corner/Endpoint” snapping mode active. Trim the vertical lines to the new horizontal line using the command **Trim > Trim Object > Length / Radius to Object.** To do this, click first on the upper part of the line to be trimmed and then on the object, until the line is the correct length.

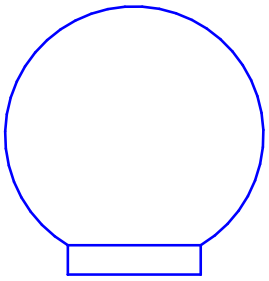


Figure 16

For the lower part of the ball head, draw a rectangle 24 mm wide and 102 mm minus 35 mm minus 38 mm (29 mm) high somewhere on the page. This is the height of the ballhead minus the height of the parts already drawn. This sum can be entered directly into the status line as simple mathematical operations are allowed here. Choose “Midpoint” snapping mode and the **Shape > Move Objects > Standard** command. Place the reference point on the middle of the rectangle’s top edge and move it to the middle of the previously drawn object’s bottom edge (Figure 17).

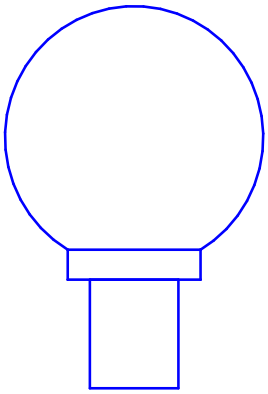


Figure 17

Use **Draw > Line > Parallel, numerical** to draw a line 2mm below the last drawn rectangle using “Edge” snapping mode to get the ends of the line in the right place (Figure 18).

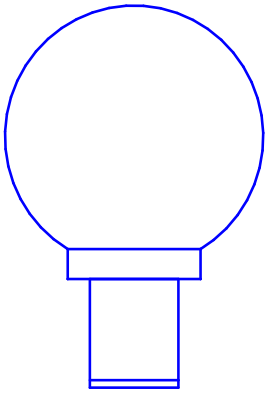


Figure 18

The next step is to create the chamfer at the bottom of the ballhead. Using **Draw > Line > Angle to Line** draw a 45° line in the corner. Turn on “Corner/ Endpoint” snapping mode when specifying the start point. When the start point has been chosen, the program will show a dialog allowing you to choose a reference line, shown during the process. This Construction Aid line lies on the previously specified line whose endpoint still has to be specified. All the intersections of this reference line can be snapped to using a combination of “Intersection” and “Construction Aid” snapping modes. Use the intersection of the reference line with the lower edge of the ballhead as the snapping point. The reference line is automatically turned off when the command has been carried out.

Carry out the same procedure for the second chamfer, but using an angle of -45° (Figure 19).

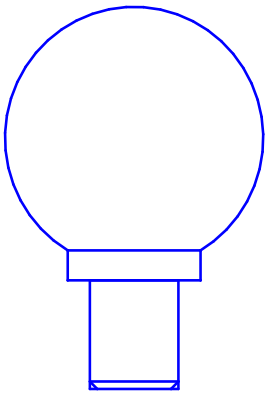


Figure 19

Now it is necessary to remove the corners next to the chamfers. Remember that these corners are actually part of the surrounding rectangle. To trim them, the rectangle must first of all be split into individual lines. Choose **Trim > Trim Object > Resolve Completely** and click on the rectangle. Before proceeding further, it is a good idea to zoom in on the area using **View > Zoom > Section**.

After choosing **Trim > Trim Object > Length / Radius to Object** click first on the end of the line to be shortened and then on the object to which it is to be shortened (Figure 20).

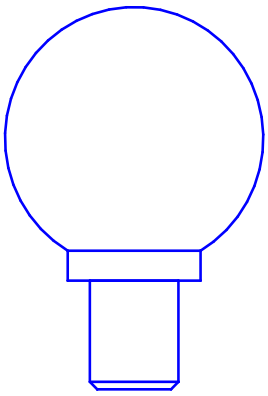


Figure 20

The construction of the blank area is more complex. In order to get some necessary intersections, the symmetry lines are drawn in at this stage. Change to the pen "0.25 mm\Dash-Dot Line Narrow" and use **Draw > Line > Center Cross** to draw the arc's center cross. Make the vertical line longer using **Trim > Trim Object > Length / Radius to Point** (Figure 21).

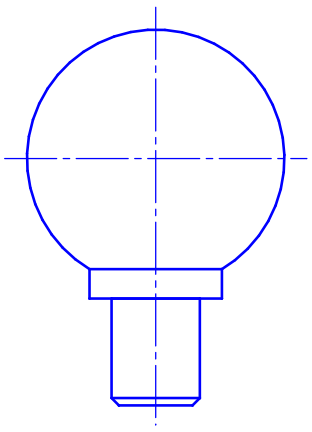


Figure 21

Change back to the pen “0.5 mm\Solid Line Wide” for the remaining steps. Using **Draw > Line > Parallel, numerical** draw a short line 24 mm above the horizontal line so as to create an intersection with the arc’s left side. Join this intersection to the center using the command **Draw > Line > Standard**). See Figure 22.

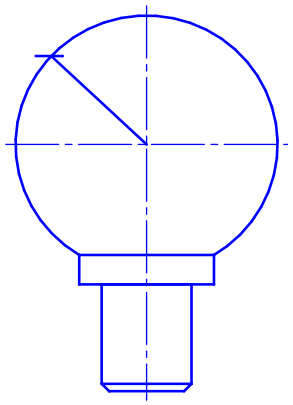


Figure 22

Remove the short horizontal line using **Shape > Delete Objects** (or by using the DEL key) and make the diagonal line longer by choosing **Trim > Trim Object > Length/Radius to Point.** Identify the lower part of the line, press F8 and enter 45 mm as the length (Figure 23).

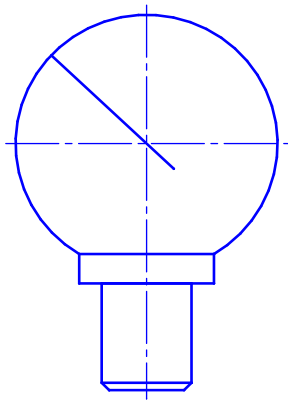


Figure 23

The edges of the blank area are generated next using the symmetry axis just drawn as a model. Choose **Shape > Move > Perpendicular** and activate the duplicate function by pressing F7 or clicking on the button in the panel. Identify the diagonal as the object to be moved and as the reference object. The reference point can be placed as desired. Press F8 and enter 10mm as the distance. (Note the direction of the arrow in the status line. It shows the direction of movement.) Repeat the process using a distance of -10 mm (Figure 24).

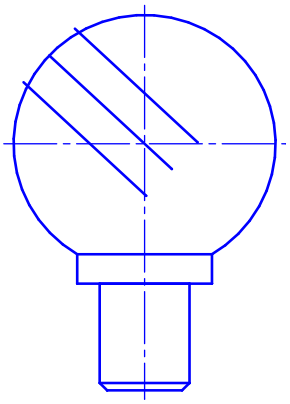


Figure 24

Use the command **Draw > Line > Standard** with the "Corner/Endpoint" snapping mode turned on, to draw a line joining the ends of the new lines. Trim the upper ends of the lines to the arc radius using **Trim > Trim Objects > Length / Radius to Object**). See Figure 25.

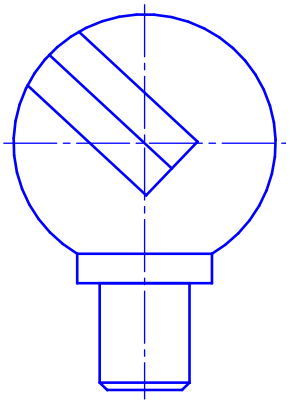


Figure 25

Now, change the pen properties of the middle diagonal line from pen “0.5 mm\Solid Line Wide” to pen “0.25 mm\\Dash-Dot Line Narrow” with **Shape > Edit Properties**. The “Fixed” check box is enabled automatically after choosing this command. Extend the blank space’s symmetry line outside the edges of the blank area using **Trim > Trim Object > Length / Radius to point** (Figure 26).

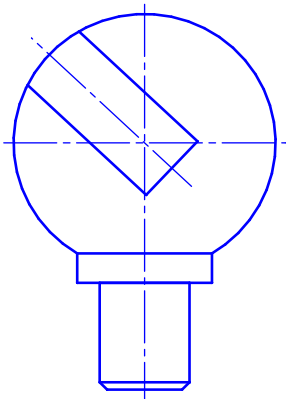
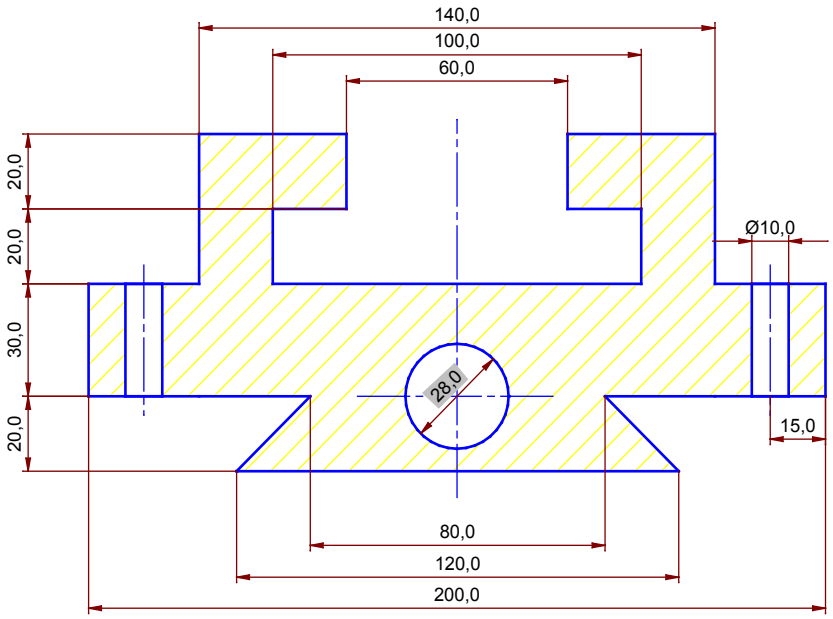


Figure 26

The rest of the procedure, such as adding dimensions and text and centering the drawing, follows the same methods as when the Construction Aid is used. Finally, save the drawing with **File > Save Drawing as**.

Using Surfaces in Construction



This chapter uses some commands that are only available in Malz++Kassner CAD6 Pro, CAD6 Studio, and/or CAD6 Industry!

Settings

`Tutorial.mkd`

Aim

This example illustrates the use of surface operations alongside the Construction Aid and trim functions when constructing a drawing. The object will be drawn solely with the addition and subtraction of surface objects and after that will have dimensions applied to it.

Surfaces

Choose A4 page size with the command **View > Pages > Edit**. In order to make the example clearer, the final shape is shown as a thin dotted line in the illustrations. This is to help make it clear which part you are drawing at any time. Whenever possible, use the F8 (ENTER) key and enter dimensions directly in the dialog.

Choose the pen “0.5 mm\Solid Line Wide” and use **Draw > Polygon > Rectangle**) to draw a rectangle 140 mm × 70 mm (Figure 1).

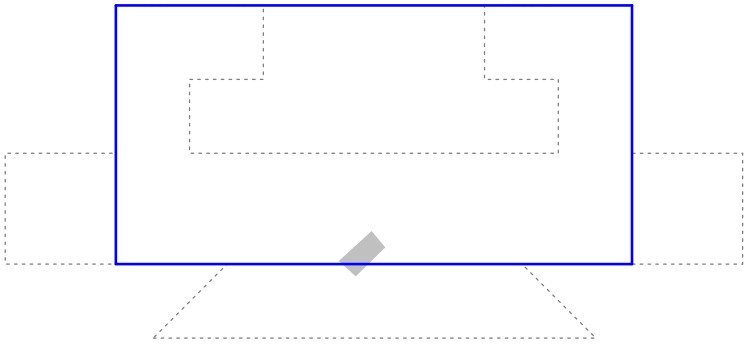


Figure 1

Draw another rectangle 200 mm × 30 mm, whose bottom left corner is at the same place as the bottom left corner of the first rectangle. Use “Corner/ Endpoint” snapping mode to ensure this. To align the two rectangles, choose **Shape > Align Objects > Centered, Frame Horizontal**. Identify the second rectangle and, using “Corner/Endpoint” snapping mode, draw a frame exactly on top of the first rectangle. The second rectangle will automatically be aligned to this frame (Figure 2).

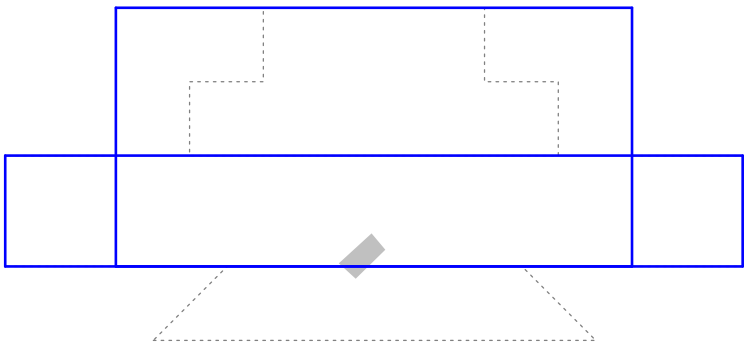


Figure 2

The two rectangles are now merged into one object using **Trim > Trim Surface / Curve > Union (A=A+B)** The two rectangles must be identified one after another. Your drawing should now look like Figure 3.

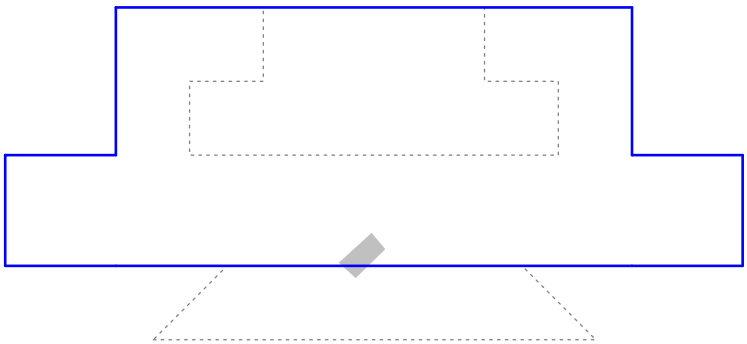


Figure 3

The next step is to add the outline of a triangle. You can see from the finished drawing that this is a right-angled triangle with a 120 mm hypotenuse (long side) (Figure 4). There are two possible ways of constructing the triangle: graphic and mathematical.

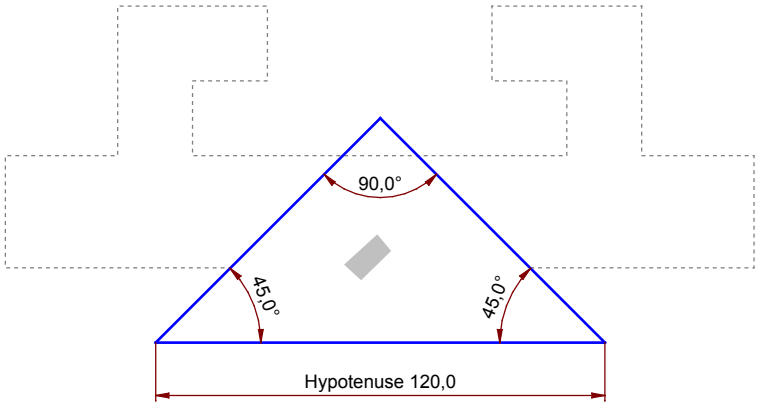


Figure 4

The Graphical Method

Draw a rectangle 120 mm × 120 mm anywhere on the page. Draw in the two diagonals as Construction Aid lines using **Construction > Construction Aid Endless Line > Standard** with "Corner/Endpoint" snapping mode active. You can determine the outline of the rectangle from this construction (Figure 5).

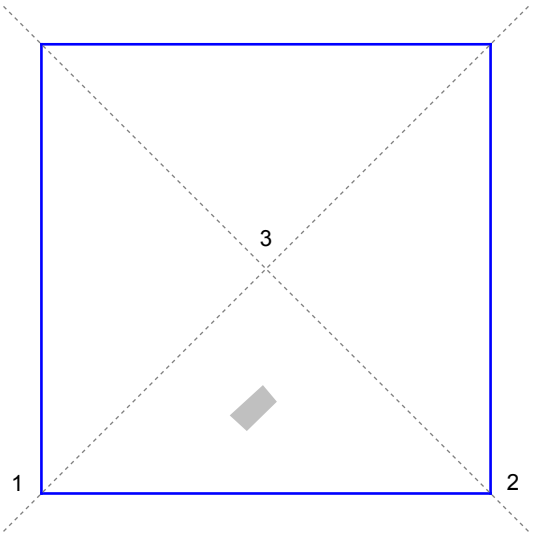


Figure 5

Choose **Draw > Polygon > Triangle** and place the corners on the two lower corners of the rectangle (1) and (2) and the intersection of the two Construction Aid lines (3). After you have drawn the triangle, you can delete both the square and the Construction Aid lines using **Shape > Delete Objects**.

The Mathematical Method

This method requires a little more knowledge about the mathematical construction of triangles, but is faster. Choose **Draw > Polygon > Triangle**. Click the mouse button to place the first point of the triangle on the page. Press the F8 key and use polar coordinates to enter the length and angle of the hypotenuse. Assuming the first point is the lower left angle, the following values have to be entered: `l=120 a=0` . This tells Malz++Kassner CAD6 to place a point 120 mm from the first one at a direction of 0°. Press ENTER and then F8 again to enter the position of the third point. The angle of the right side A and its length are calculated using Pythagoras" formula: The height is identical to half the hypotenuse C, as can be seen from Figure 6.

$$A = \sqrt{h^2 + \left(\frac{C}{2}\right)^2}$$

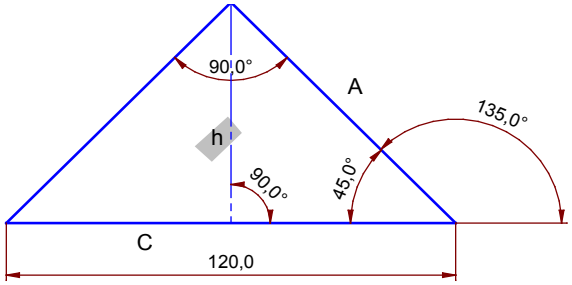


Figure 6

In the status line enter `l=sqrt(60^2+60^2) a=135` .

Back to the Outline

To align the triangle correctly with the existing outline, draw a horizontal Construction Aid line 20 mm below the bottom line of the outline. Do this by choosing **Construction > Construction Aid Endless Line > Parallel, numerical** and entering 20 mm as the distance. Use **Shape > Move Objects > Standard** and activate "Edge" snapping mode to move the triangle onto the new line. The triangle is horizontally aligned with the outline using the **Shape > Align Objects > Centered, Frame Horizontal** in the same way as the two rectangles were aligned earlier.

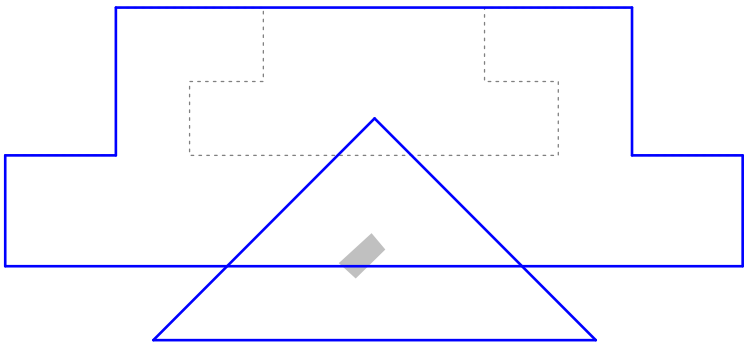


Figure 7

These two outlines are also merged with the command **Trim > Trim Surface / Curve > Union(A=A+B)**.

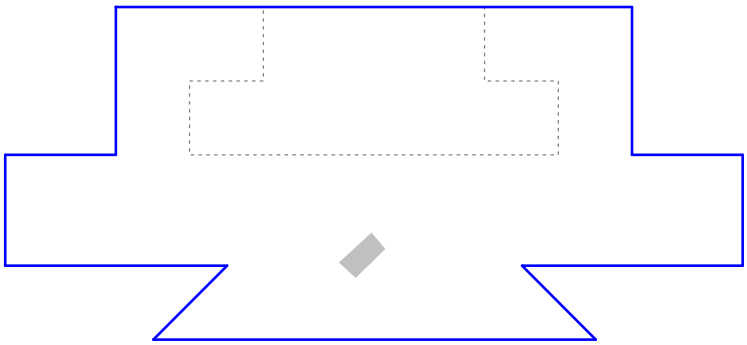


Figure 8

Further areas have to be removed from the outline. Draw a rectangle 100 mm × 20 mm with its bottom left corner on outline corner 1 (see Figure 9).

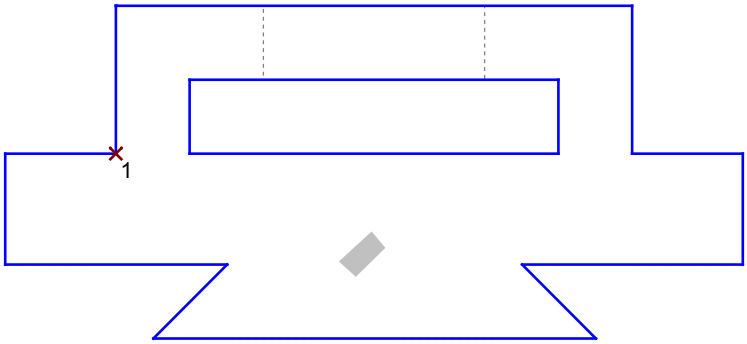


Figure 9

This rectangle has to be aligned horizontally using **Shape > Center Objects > Frame, Horizontal**). Use the two symmetrical points in the outline as the corner points of the frame. Use **Trim > Trim Surface / Curve > Difference (A=A-B)** to unify the objects. Identify both objects one after another. A hole appears in the place of the last-drawn rectangle. This change is not visible if the larger object is filled.

You can easily check the alterations to the outline by choosing **Shape > Edit Properties** and changing the fill mode to “both”. You will see the blank space. Objects behind the outline will now be visible through the hole.

One further rectangle is to be removed. Draw a 60 mm × 40 mm rectangle and position it using **Shape > Move Objects > Standard** so that it covers areas approximately the same are above and below the upper part of the outline. Center the rectangle horizontally using **Shape > Align Objects > Centered, Frame Horizontal**).

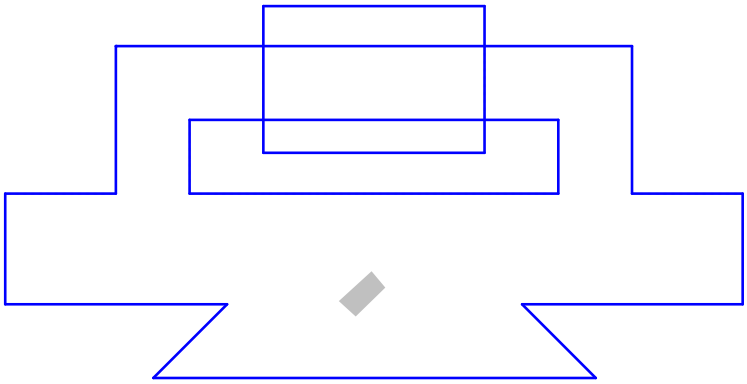


Figure 10

Use **Trim > Trim Surface / Curve > Difference (A=A-B)** to unify the objects (Figure 11).

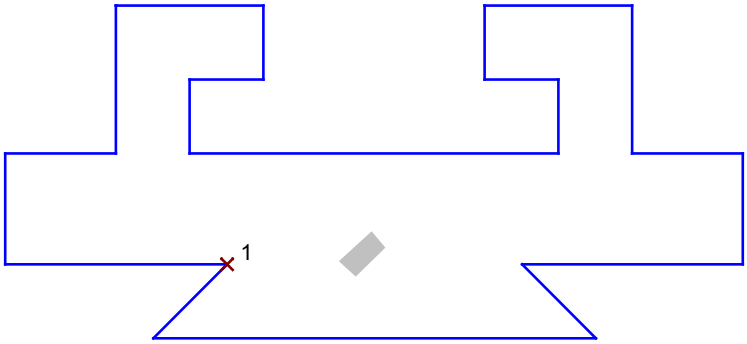


Figure 11

Use **Draw > Circle > Standard** to draw a circle with a radius of 14 mm and its center at outline corner point 1 (Figure 11). Center the circle horizontally using **Shape > Align Objects > Centered, Frame Horizontal**. This object is also cut away using the command **Trim > Trim Surface / Curve > Difference (A=A-B)** (Figure 12).

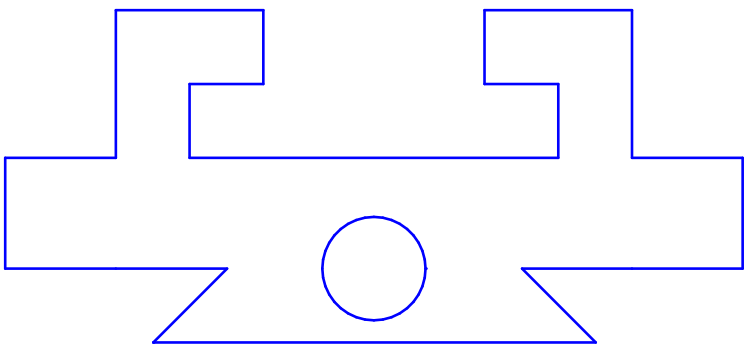


Figure 12

Draw a further rectangle 10 mm × 30mm and align it centrally using **Shape > Align Object > Centered, Frame Both** between the points 1 and 2 (Figure 13).

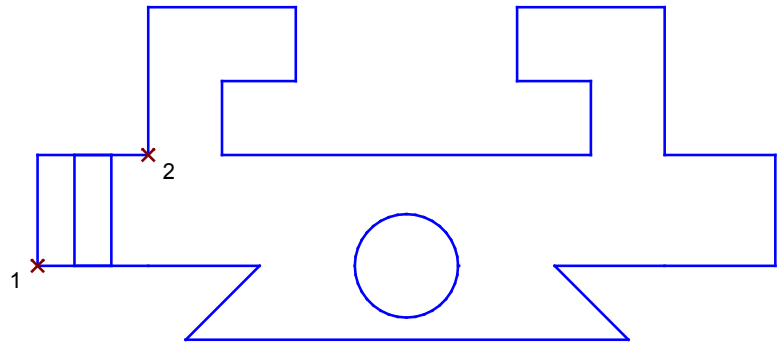


Figure 13

You can now use **Shape > Move Object > Relative**. to mirror the positioned rectangle about the center. Enable the "Duplicate" check box in the dialog and enter a value for the copy position of 170 mm horizontally.

The command **Shape > Reflect Object > Line** cannot be used because an axis would have to be identified and there is no axis present.

These two rectangles also have to be merged with the larger outline. However a difference does not have to be created, but they have to be combined with the large outline. This will keep the upper and lower outlines intact. Choose **Trim > Trim Surface / Curve > Combine Outlines** and identify all the objects to be combined. Either hold down the CTRL key and click on each object in turn or hold down the SHIFT key and drag a rectangle around all the objects. The result of this operation will not be immediately visible, however you can test it (as previously) by coloring the object using **Shape > Edit Properties** (Figure 14).

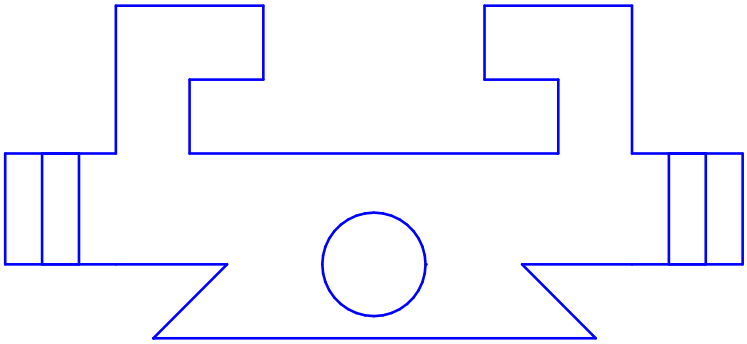


Figure 14

Choose the pen “0.25 mm\Solid Line Narrow”. Hatch the whole outline using the command **Draw > Hatching > Objects** (Figure 15).

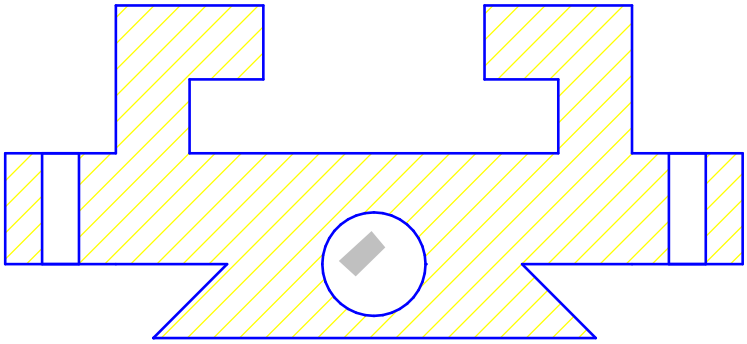


Figure 15

Now the symmetry lines in the circle and the cut-throughs at left and right should be added. Click on the pen “0.25 mm\\Dash-Dot Line Narrow” and choose **Draw > Line > Center Cross** and then snap the circle’s center-point to draw a center cross which extends beyond the boundary of the circle. Use **Trim > Trim Object > Length / Radius to Point** to extend the vertical line upwards to roughly level with the object’s upper edge.

In the cut-out on the left, turn on “Midpoint” snapping mode and use **Draw > Line > Vertical** to draw a vertical line. Use **Trim > Trim Objects > Length / Radius to Point** to extend the line upwards by 5 mm. To do this, press the F8 key after identifying the line and enter 35 mm (30 mm existing line, 5 mm extension. Repeat the process with the other end, entering 40 mm (35 mm line plus 5 mm extension).

Copy this vertical line to the other cut-out by using **Shape > Reflect Object > Line** with the duplicate function (F7) turned on. Identify the line to be copied and then click on the vertical line running through the circle which is the whole outline’s symmetry axis. You could also use the command **Shape > Move Objects > Relative** (Figure 16).

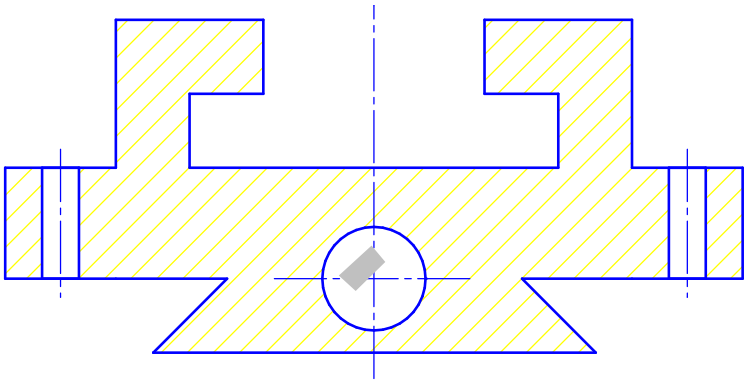


Figure 16

Finally add the dimensions. Apart from the circle diameter these can all be added using **Dimension > Dimension > Length, Point** or **Dimension > Dimension > Length, Objects** To be able to position the diameter dimension better, after choosing **Dimension > Dimension > Diameter, Object** turn off the “Close” and “Center” settings In the dimension parameters dialog (accessible by pressing SHIFT+ESC).

Place an erasing surface behind the hole’s dimension and remove the diameter sign from this dimension using **Shape > Edit Text**). Your drawing should now look like Figure 17.

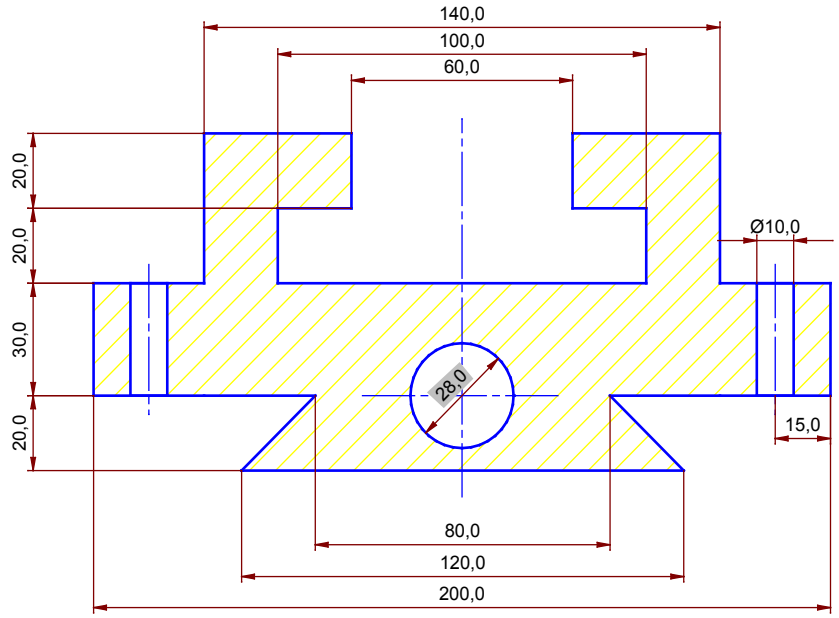
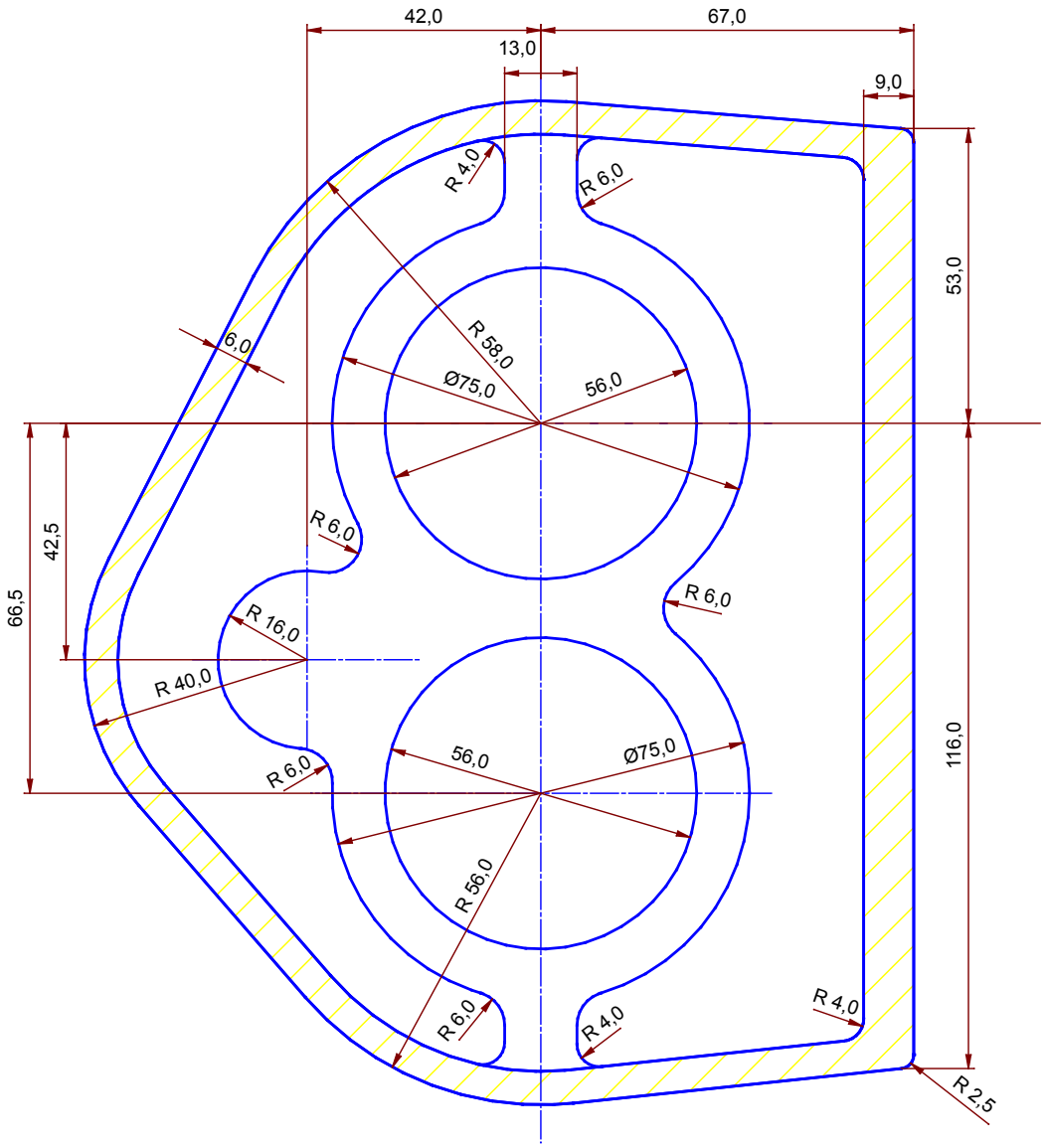


Figure 17

Do not forget that there can be problems using **Trim Surface / Curve > Union**, **Trim Surface / Curve > Intersection** and **Trim Surface / Curve > Difference** if adjacent surface edges are parallel and lie on top of each other. If possible, avoid this by making objects overlap. Further information about this can be found in the electronic reference.

Gear Housing



This chapter uses some commands that are only available in Malz++Kassner CAD6 Pro, CAD6 Studio, and/or CAD6 Industry!

Aim

Drawing the gear housing should illustrate generating complex and nested surfaces. Layers are used in order to keep the drawing manageable. Layers can be used to make parts of the drawing which have already been finished invisible.

Settings

Again, the settings contained in `Tutorial.mkd` are used.

The drawing will be done on an A4 portrait page, which means that the page settings do not have to be changed. Extra layers will, however, be needed. Choose **View > Layers > Edit Layers** and alter the name of layer “Drawing” to “Inner Housing Wall” (using the “Name...” button) and then click on “New” and create a new layer with the name “Outer Housing Wall”. Reselect the layer “Inner Housing Wall” and leave the dialog with the button “Activate“.

Begin the drawing with two Construction Aid lines running horizontally and vertically through the center of the page (**Construction > Construction Aid Endless Line > Horizontal** and **Construction > Construction Aid Endless Line > Vertical**). Direct input of coordinates using F8 (ENTER) ensures exact positioning. For the horizontal line enter the y-coordinate `(_pb+_pt)/2` , for the vertical line enter the x-coordinate `(_pl+_pr)/2` .

The centers of the inner housing circles are 66.5 mm apart. To determine the center positions precisely, add parallel Construction Aid lines 33.25 mm above and below the horizontal Construction Aid line. The intersections of these lines with the vertical Construction Aid line mark the center points of the circles. After choosing the pen “0.5 mm\Solid Line”, activate “Construction Aid” snapping mode to snap to the intersections and draw two circles (**Draw > Circle > Standard**) with a diameter of 56 mm (Figure 1).

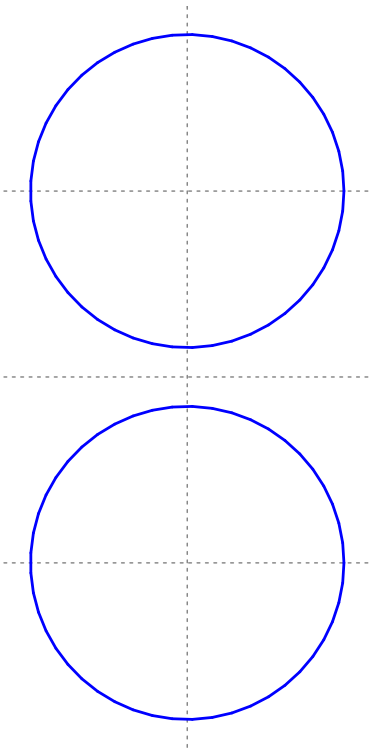


Figure 1

Two Construction Aid circles with a radius of 75 mm to mark the outline around the circles are now added using **Construction > Construction Aid Circle > Concentric** and entering the new radius using F8 (ENTER) (Figure 2).

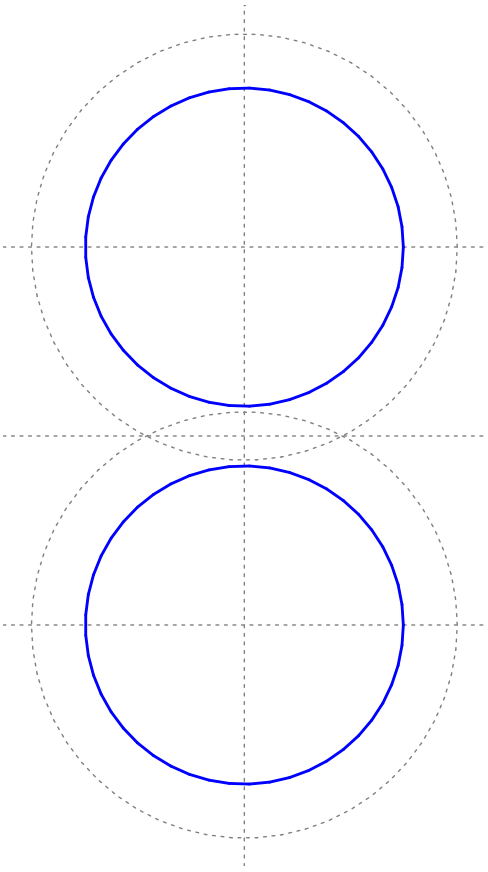


Figure 2

To work out the center position of the third circle which makes up part of the outline, add a parallel Construction Aid line 42.0 mm to the left of the existing vertical Construction Aid line (**Construction > Construction Aid Endless Line > Parallel, numerical**). A further horizontal Construction Aid line 42.5 mm below the horizontal Construction Aid line running through the center of the upper circle determines the center of the third circle part. Draw a Construction Aid circle with a radius of 16 mm from this point. In addition you can add two parallel Construction Aid lines 6.5 mm on either side of the vertical center Construction Aid line. These two lines mark the distance between the inner and outer walls (Figure 3).

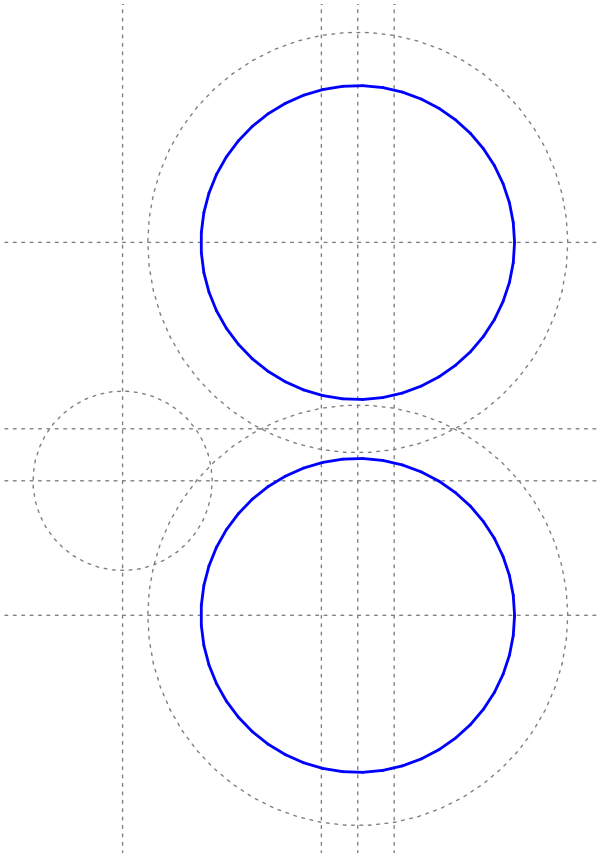


Figure 3

The Construction Aid circles are joined using arcs with a radius of 6 mm. Add these arcs to the drawing using **Draw > Circular Arc > Radius - Object - Object**. After entering the radius identify the relevant Construction Aid circles and click on the “Forwards” key in the “Choose object” window until the arc you want to add appears on screen, and then click on OK.

Repeat the process until all the arcs have been added (Figure 4).

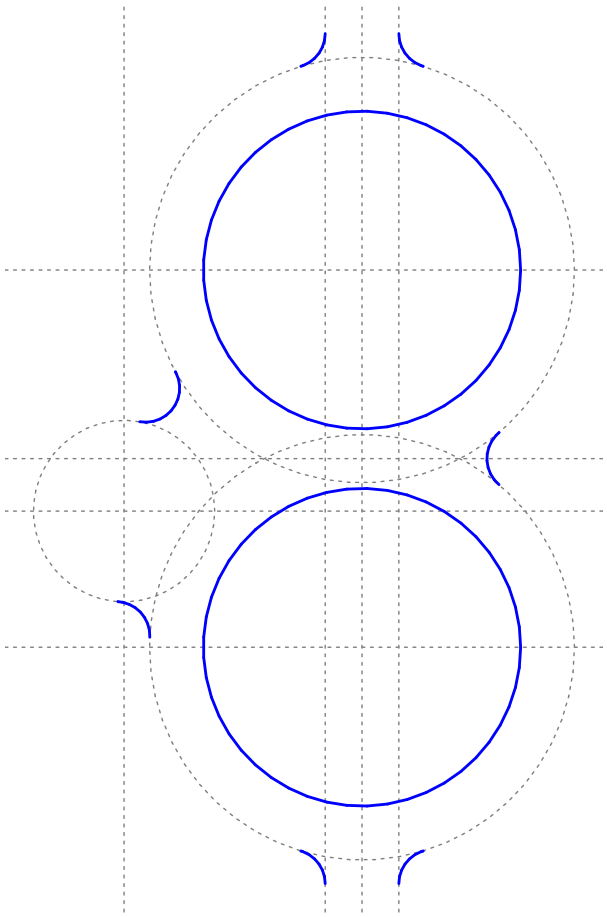


Figure 4

Add the missing circular arcs between the 6 mm connecting arcs using the command **Draw > Circular Arc > On Circle**. To do this, identify each of the relevant Construction Aid circles in turn and snap to the ends of the existing circular arcs using “Corner/ Endpoint” snapping mode (Figure 5).

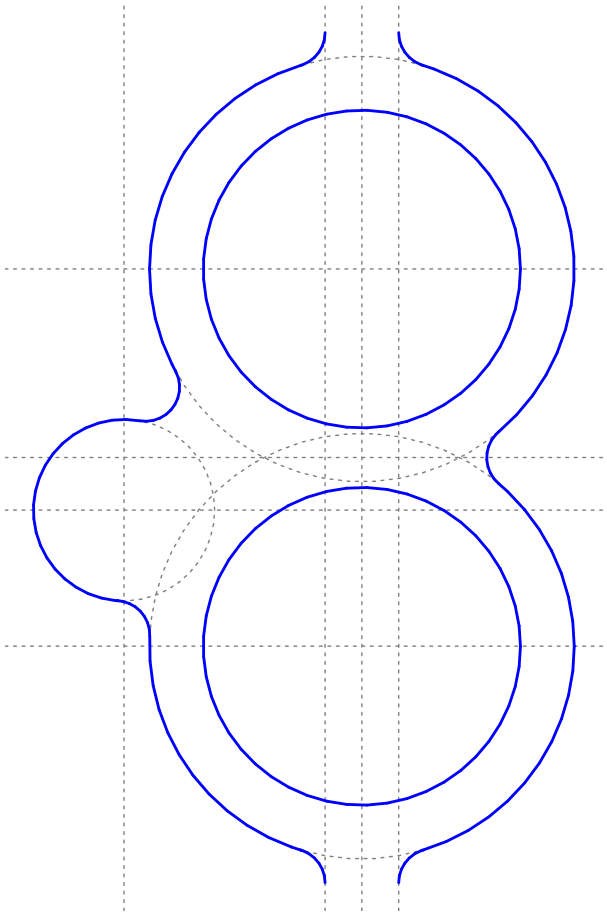


Figure 5

In order to be able to construct the correct transition from the inner to the outer wall, draw a Construction Aid circle concentric with the upper large Construction Aid circle with a radius of 52 mm and a Construction Aid circle concentric with the lower large Construction Aid circle with a radius of 50 mm (**Construction > Construction Aid Circle > Concentric**). This value is calculated from the radii of the outer wall outline (56 mm / 58 mm) minus the thickness of the wall (6 mm) (see Figure 6).

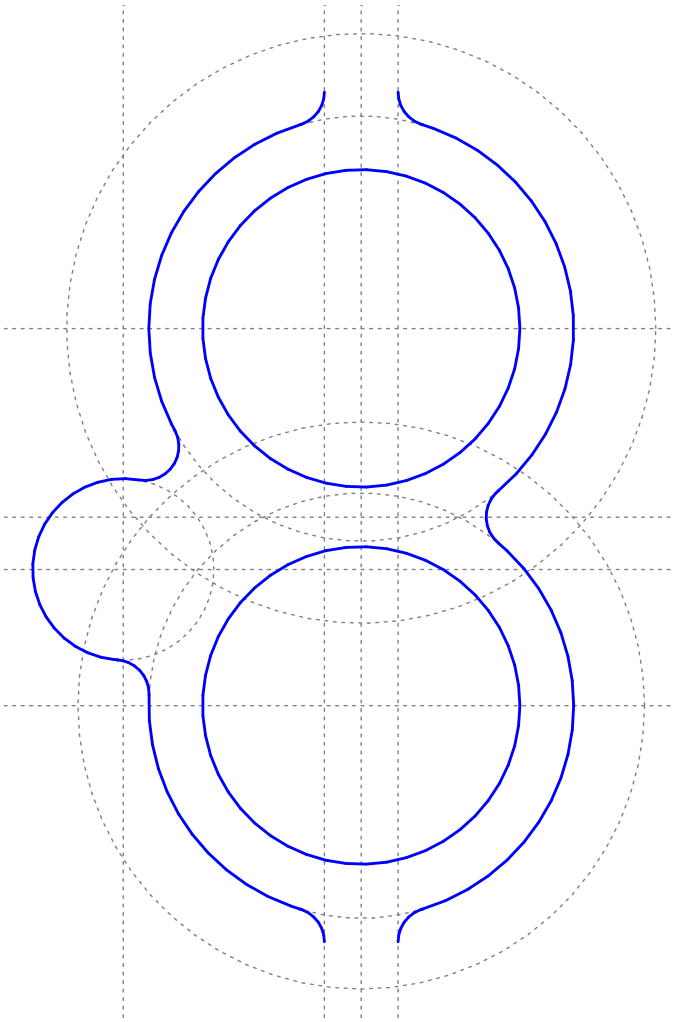


Figure 6

The missing circular arcs with a 4 mm radius are added next, again using **Draw > Circular Arc > Radius -Object - Object**. The circular arcs are tangential to the just-constructed Construction Aid circles and the vertical Construction Aid lines 6.5 mm from the central vertical Construction Aid line. Activate “Corner/ Endpoint” snapping mode and join the 4 mm arcs to the relevant 6 mm arcs using **Draw > Line > Standard**). (Figure 7).

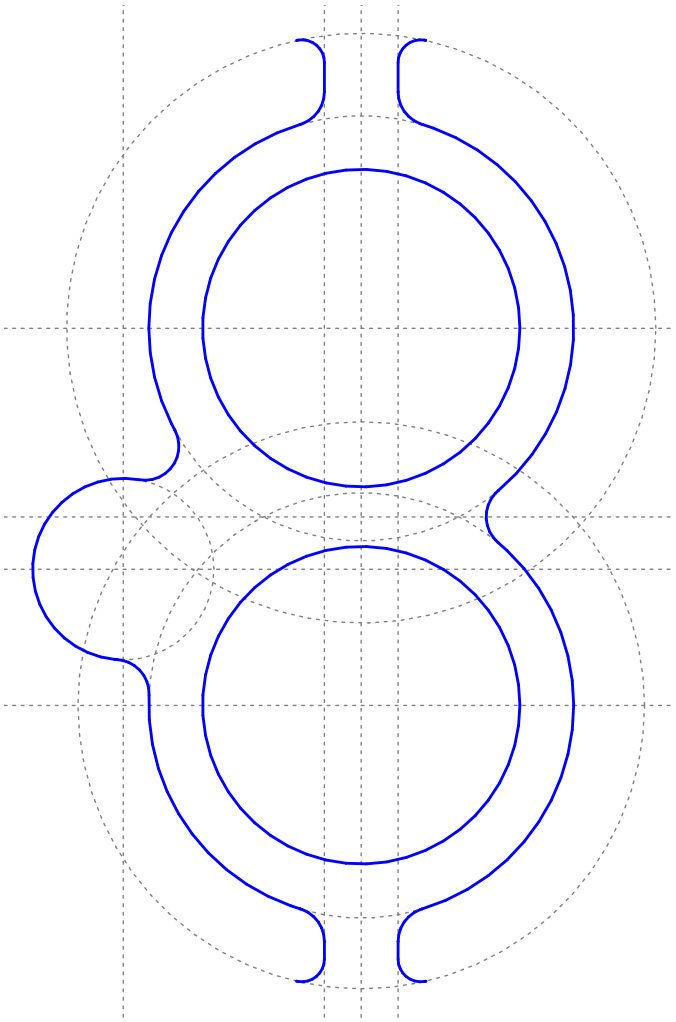


Figure 7

This completes the inner housing wall. You can now remove all of the existing Construction Aid to make it easier to get an overview of the drawing. To do this, choose **Construction > Delete Completely**.

To keep a reference for the rest of the drawing, use the command **Construction > Construction Aid Endless Line > Center Cross** and snap circle A's center-point to draw its center cross. In addition, draw Construction Aid circles with a radii of 58 mm concentric to circle A, 56 mm concentric to circle B and 40 mm concentric to circle C (see Figure 8).

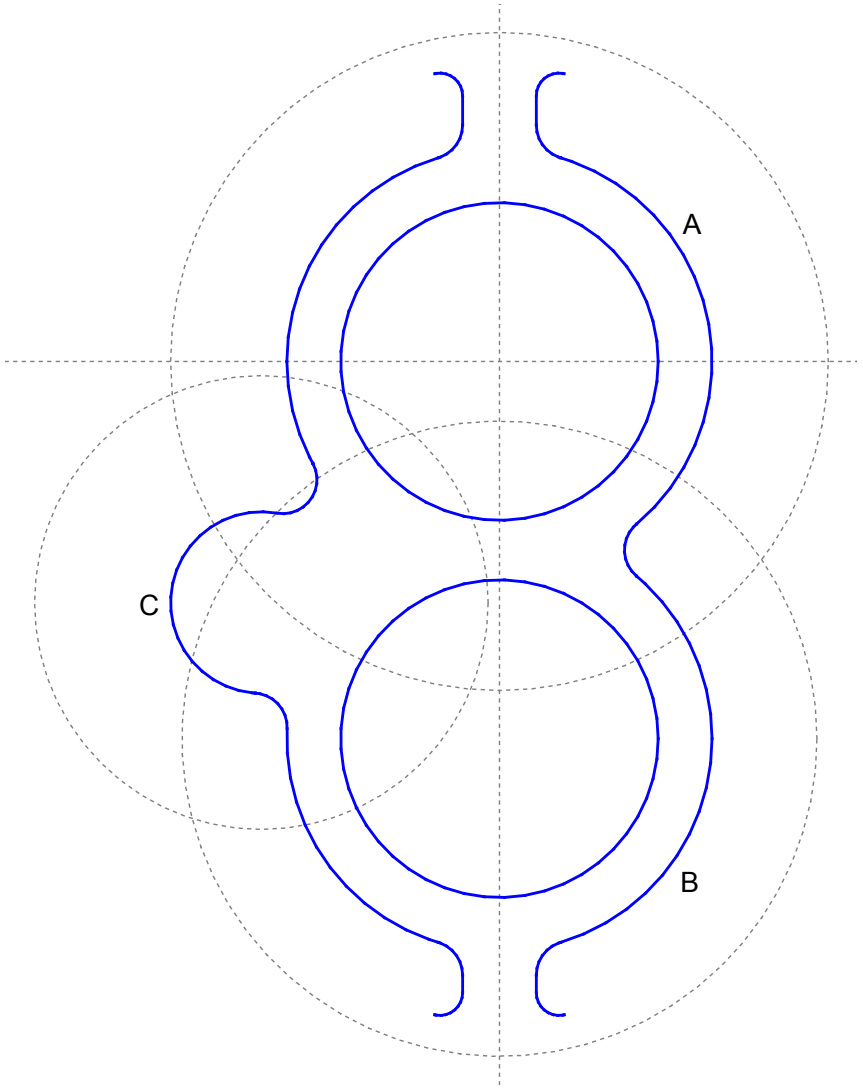


Figure 8

It is not necessary to display the inner housing wall while constructing the outer outline. Choose **View > Layers > Edit** and in the dialog which appears choose the "Inner housing wall" layer. Deactivate the "Display" check box in the "Selected Layer" section. Make "Outer housing wall" the active layer and click on "Activate". The inner housing wall is now hidden from view and will not get in the way when working on the outer housing wall.

Lines must now be added to the Construction Aid. Using **Construction > Construction Aid Endless Line > Parallel, numerical** draw Construction Aid lines 67 mm to the right of the central vertical line, 53mm above and 116 mm below the central horizontal Construction Aid line (Figure 9).

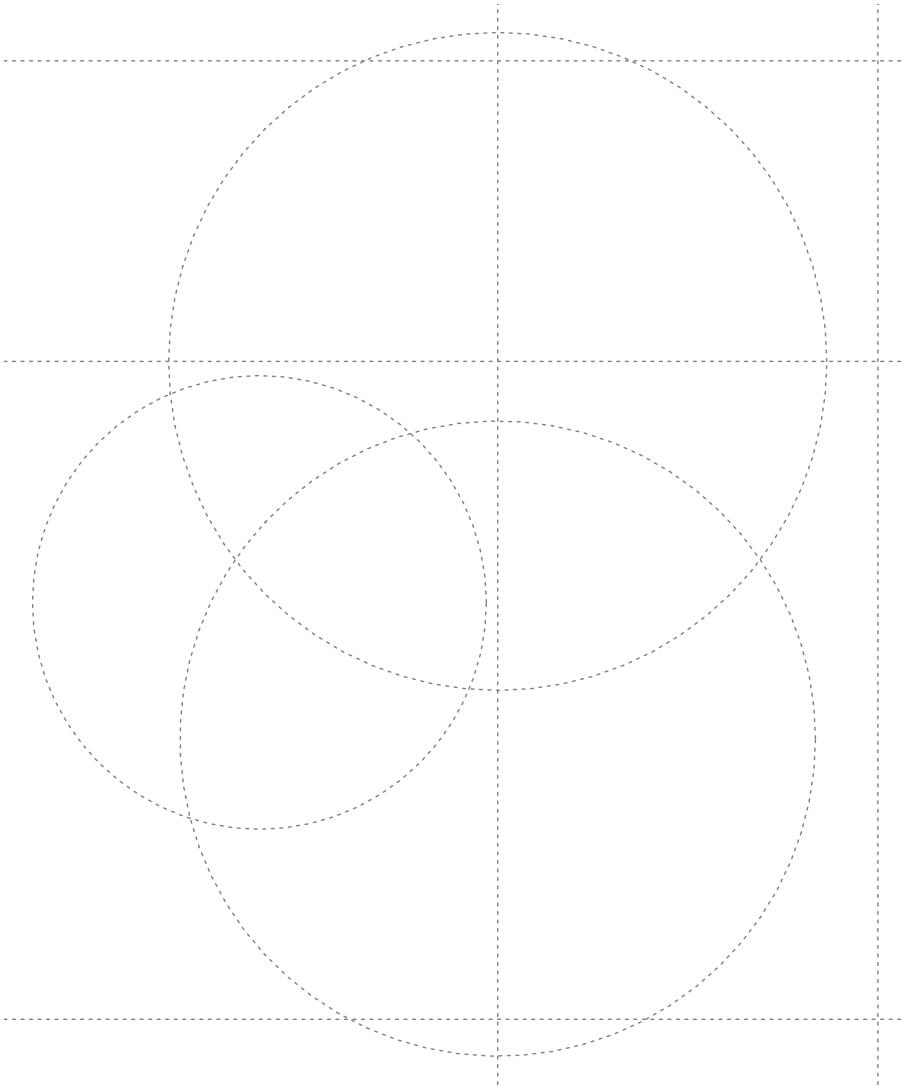


Figure 9

To describe the outer outlines of the wall, the roundings at the top and bottom on the right edge are missing. Their radii and positions tangential to the Construction Aid lines are known. You can add the circle using **Construction > Construction Aid Circle > Radius - Object - Object** and a radius of 2.5 mm. This is shown in Figure 10.

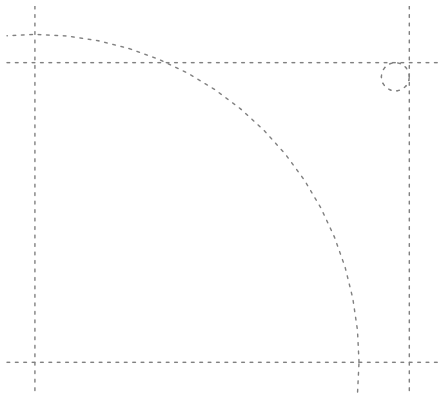


Figure 10

The outer outline consists of circular arcs and tangents to the arcs. Next, because the start and end points of the circular arcs are not known, draw the tangents to the Construction Aid circles which describe the outline. Choose **Draw > Tangent > Object - Object** and identify, one after another, both points on two circles which the line should be tangent to. You can see from Figure 11 which Construction Aid circles the lines should be tangential to.

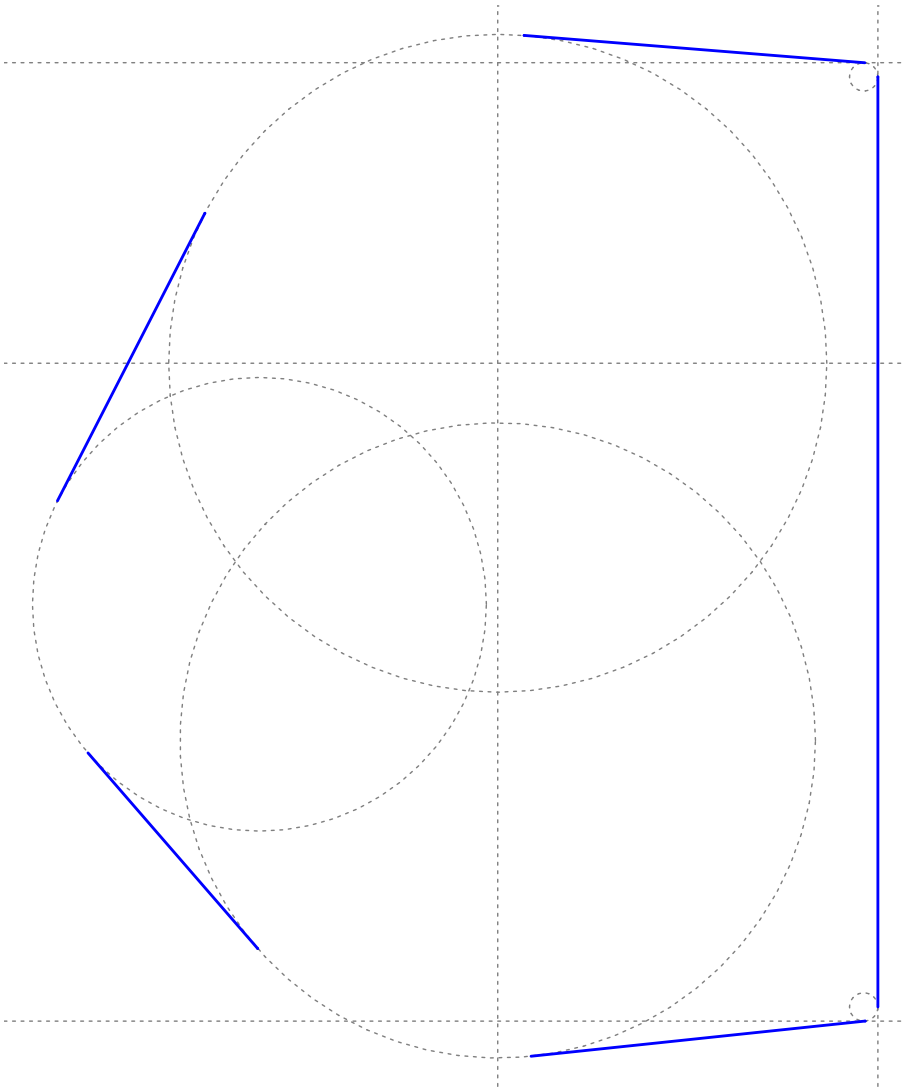


Figure 11

It is a good idea to work with various zoom levels to place the tangent points accurately on the last drawn small circle and to view things clearly. Use **View > Zoom > Section**.

Using **Draw > Circular Arc > On Circle** with “Corner/ Endpoint” snapping mode active to draw the missing circular arcs and create a closed outline (Figure 12).

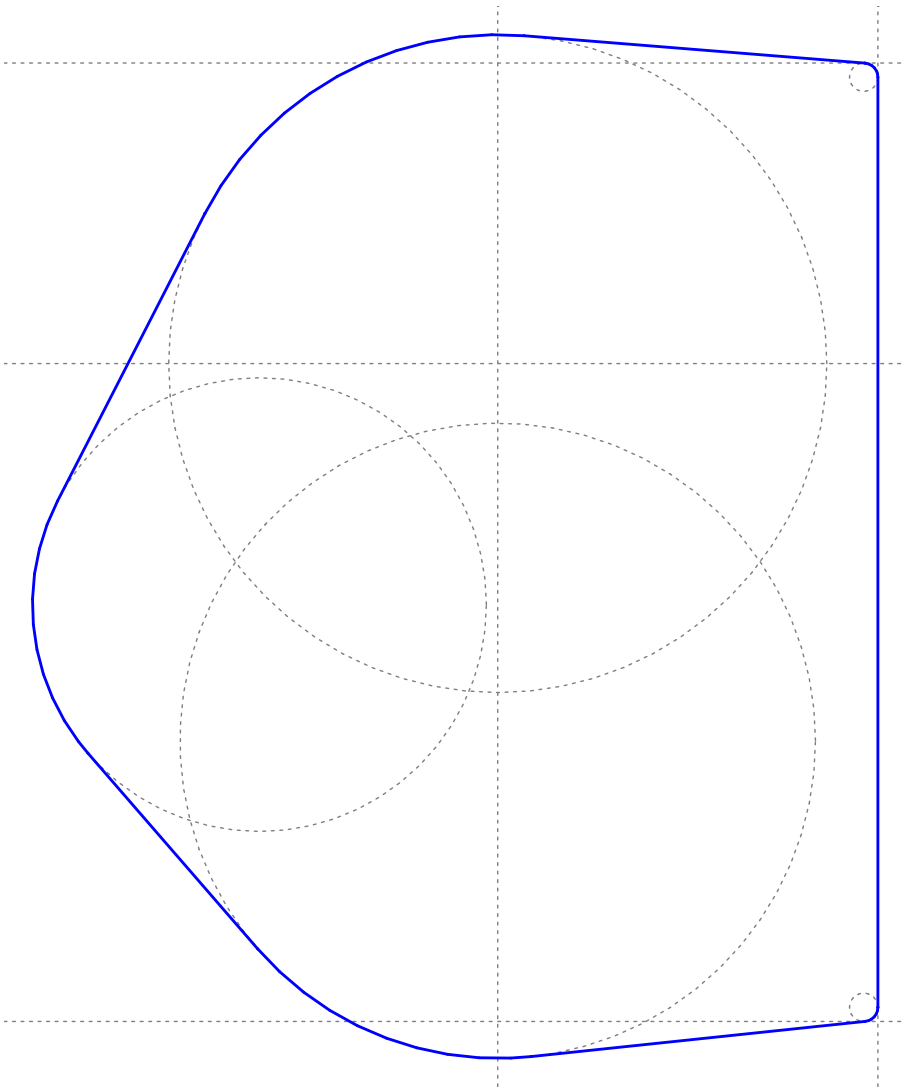


Figure 12

To construct a housing wall with a thickness of 6 mm, choose **Draw > Line > Equidistant**. Enter 6 mm in the dialog which appears. To draw an equidistant line, first of all identify the reference element and then specify which side of the reference element the equidistant line should run. Do this for all of the elements which require it. The exceptions are the two circular arcs on the right and the right-hand vertical line between them. Draw an equidistant line 9 mm to the inside of this line (Figure 13).

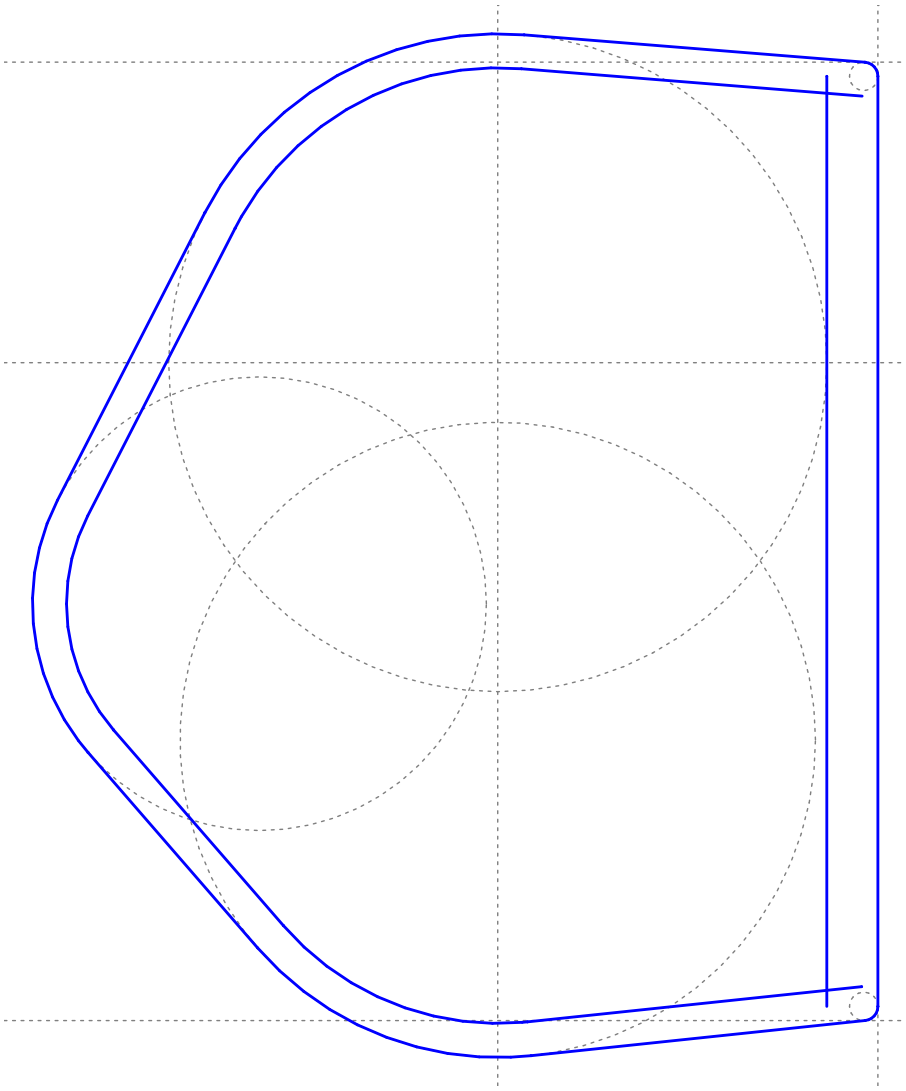


Figure 13

In the area where the 9 mm equidistant line and the 6 mm equidistant line intersect, both lines have to be rounded with a 4 mm radius. This is done using **Trim > Round Edge, Outer Arc > Object - Object**. Enter 4 mm into the dialog and identify both lines at the end to be rounded (Figure 14).

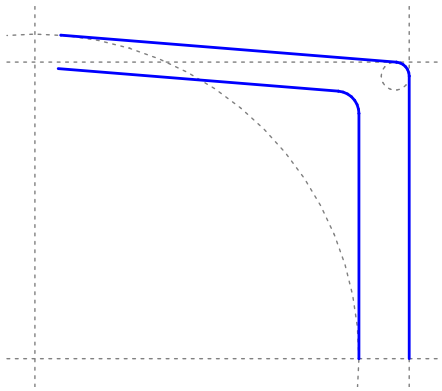


Figure 14

This completes drawing of the gear housing wall. In order to hatch it, choose the **Draw > Hatching > Generated Surface** command.

Now, identify all the visible objects. Either press F10 (SHIFT+Q) or press the SHIFT key and drag a rectangle round all the objects. For each contour to be recognized, a reference point has to be placed near to it. So place one reference point close inside the inner housing wall, and one point close outside the outer housing wall. Then terminate the point entry (click right mouse button, and if the Workflow Manager is active and appears, choose **Finish Command**). The area between the two housing walls will now be hatched (see figure 15).

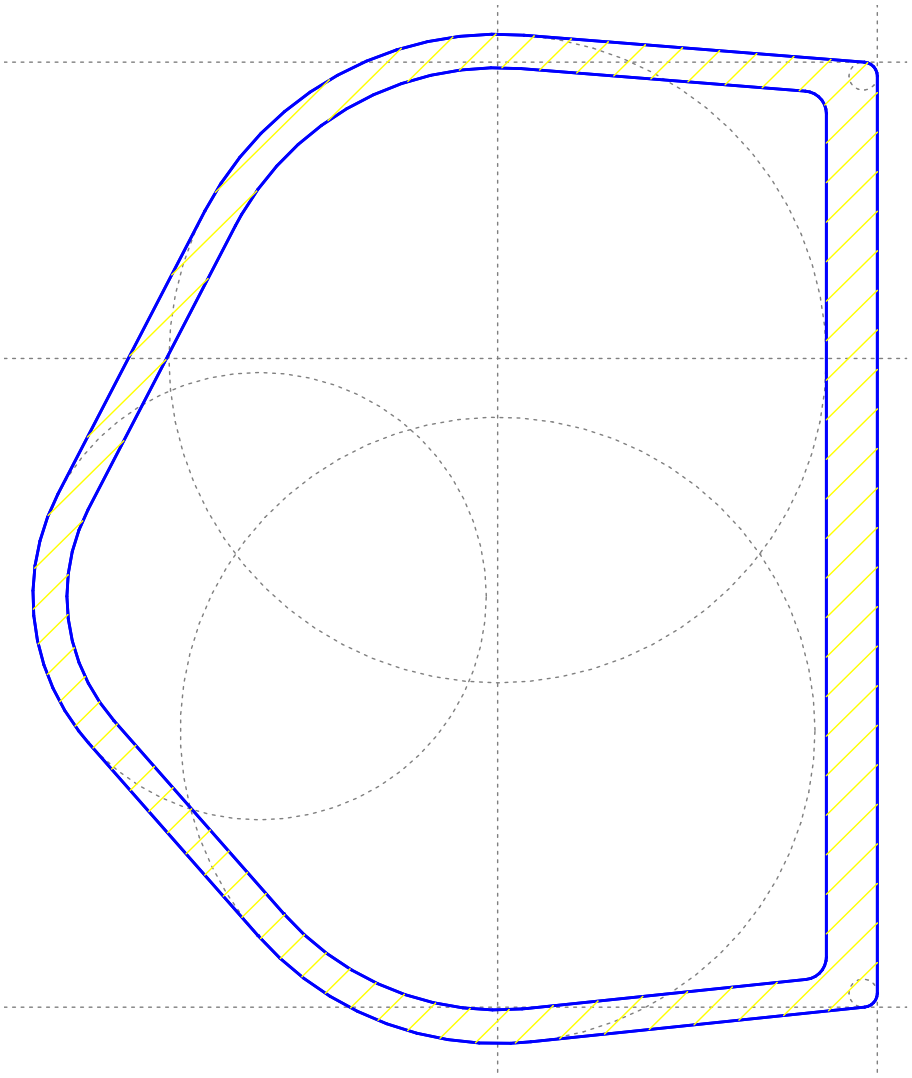


Figure 15

The Construction Aid can now be turned off using the button in the Panel (Figure 16). Then, use the **View > Layer > Edit** command to display the layer “Inner housing wall” again.

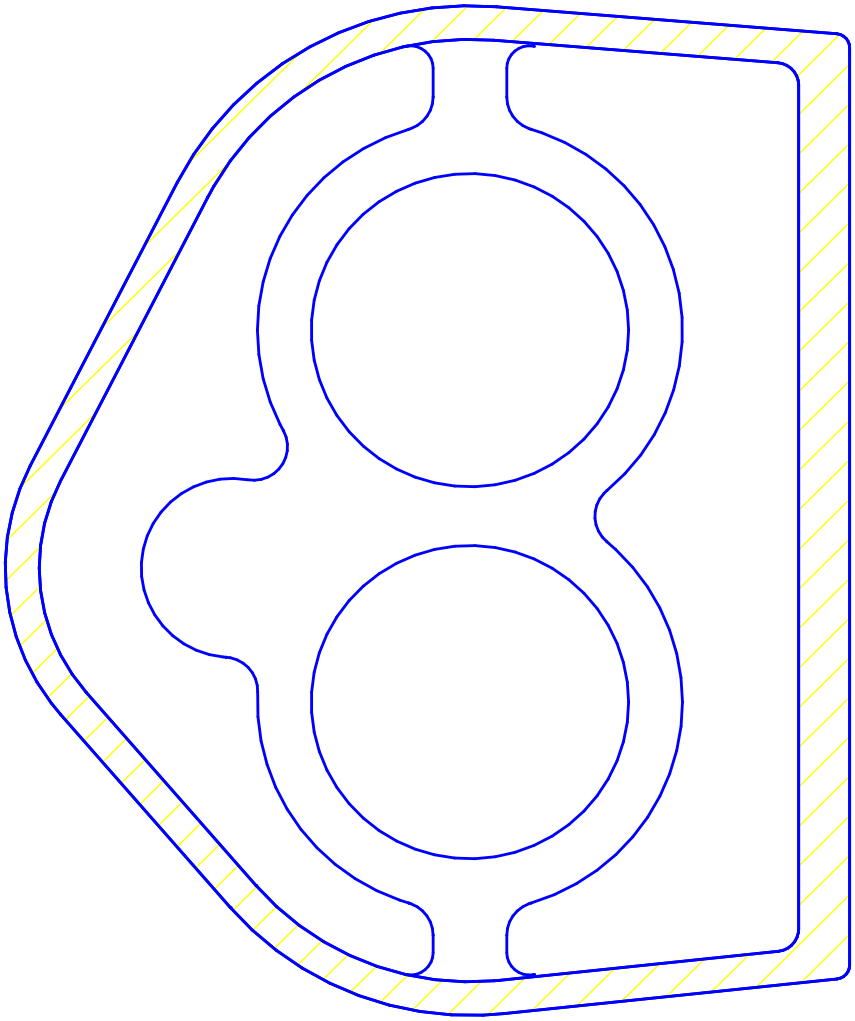


Figure 16

Remember that the hatching is on a layer of its own and can be turned on and off. Layer settings for specified objects are set using **View > Layers > Defaults**.

Next, draw the circles’ symmetry lines with **Draw > Line > Center Cross** using the pen “0.25 mm\\Dash-Dot Line Narrow”. Extend the larger circle’s vertical symmetry line outside the outer housing wall using **Trim > Trim Object > Length / Radius to Point**.

Dimension centering should be turned off for all the following dimensions by clearing the appropriate check box in the dialog. In addition, reduce the text size for all dimensions to 3.5 mm with the **Lettering > Dimension Parameters** command. Dimensions are applied to the radii using **Lettering > Dimension > Radius, Object.** The diameters can be measured by choosing **Lettering > Dimension > Diameter, Objects.** The distances between the walls are best measured using **Lettering > Dimension > Distance, Object - Object.** The remaining lengths can be measured using **Lettering > Dimension > Length, Points.**

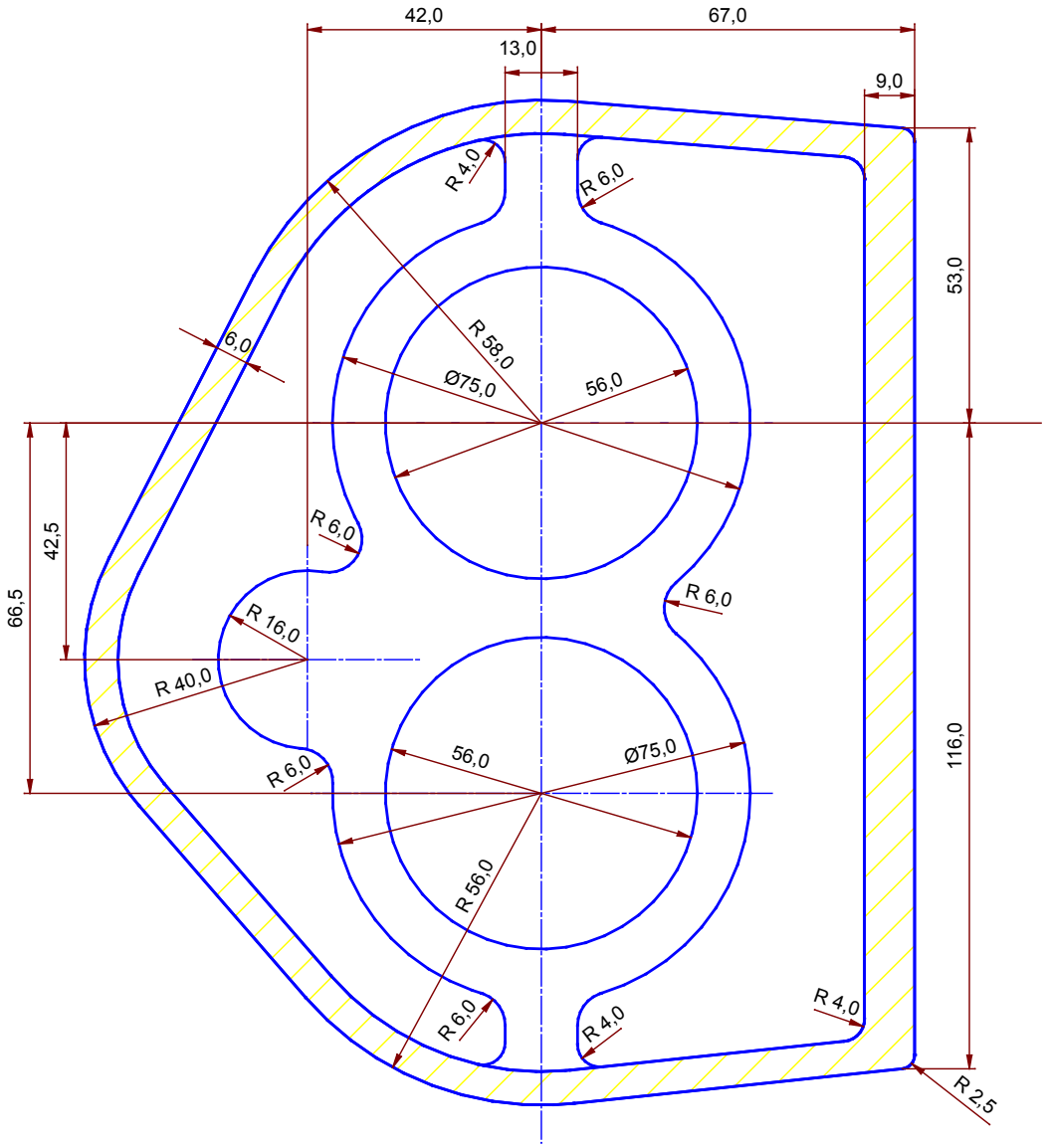


Figure 17

Choose **Shape > Edit Text** to remove unwanted diameter symbols and then add a drawing frame and text fields. Finally, save the drawing with **File > Save Drawing as.**

Tables and Parts Lists

Linetype	*Linewidth / Linegroups*				
Solid line wide	0,25	0,35	0,5	0,7	1,0
Dash-dot line wide	0,25	0,35	0,5	0,7	1,0
Dash line	0,18	0,25	0,35	0,5	0,7
Solid line narrow	0,13	0,18	0,25	0,35	0,5
Dash-dot line narrow	0,13	0,18	0,25	0,35	0,5
Freehandline	0,13	0,18	0,25	0,35	0,5

This chapter uses some commands that are only available in Malz++Kassner CAD6 Pro, CAD6 Studio, and/or CAD6 Industry!

Aim

This example should explain how to generate parts lists and tables. You will get to know the various methods of making multiple copies and how to align text into predetermined areas. In addition it goes into the process of exporting graphics for use in other applications (e.g. word processors). The example is the table which you already know from the introduction to the chapter on the pen and layer concept. Generating parts lists and text boxes for technical drawings follows a similar procedure.

Settings

`Tutorial.mkd`

Basics

The procedure for producing a table is determined by the type of table. Parts lists for technical drawings make use of a predetermined grid, tables in technical documentation usually cannot exceed certain dimensions. This example has an inner frame measurement of 140 × 70 mm. This is a precondition.

Begin with the pen “0.25 mm\Solid Line Narrow” and use **Draw > Polygon > Rectangle** to draw a rectangle 140×70 mm. Use **Trim > Trim Object > Resolve Completely** to split the rectangle into individual lines.

Choose **Construction > Multiple Copy > Object**. This makes it possible to copy a specified object to the outline of another object. In the example, seven equally spaced horizontal lines are to be added. This means that the top horizontal line is to be copied seven times in the vertical distance. Identify the top line, and with “Corner/Endpoint” snapping mode

on, place a reference point at the start point of the top horizontal line. Identify the vertical line as a reference object. Choose “linear” in the dialog and enter seven as the value. Close the dialog with OK. Seven horizontal fields should appear on the page (Figure 1).

Figure 1

The process used means that there are eight copies of the original object in the specified area. This also means that copies are placed at the start and end points of the specified reference object. However, there are already lines there (the top and bottom edges of the rectangle). To avoid being continually asked by the program which line you wish to use (e.g. during identification), remove one of the lines from the top and bottom using **Shape > Delete Objects** or by using the DEL key. Identify one of the top lines and answer “OK” to the program’s “Choose Objects” question. Repeat the process for the bottom line.

To generate the vertical lines on the right hand side, use **Construction > Multiple Copy > Length**. Identify the right hand line and specify the corner point A (with “Corner/Endpoint” snapping mode active) as the reference point. The start point also lies at point A and the end point is the mid-point M (specified with “Midpoint” snapping mode active) of the upper horizontal line. Five sections should be created (six lines); once again, there are two lines on top of each other at the left, one of which has to be deleted (Figure 2).

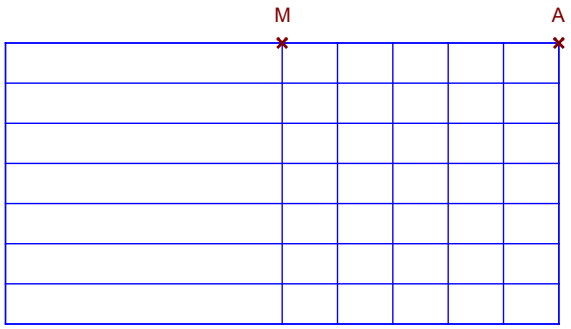


Figure 2

To shorten the column dividing lines in the top section to 2mm, add a Construction Aid line whose intersections with the vertical lines will be used later as trim points. Choose **Construction > Construction Aid Endless Line > Parallel, numerical** and enter a distance of 2 mm. Identify the second horizontal line from the top and place the Construction Aid line above it (Figure 3).

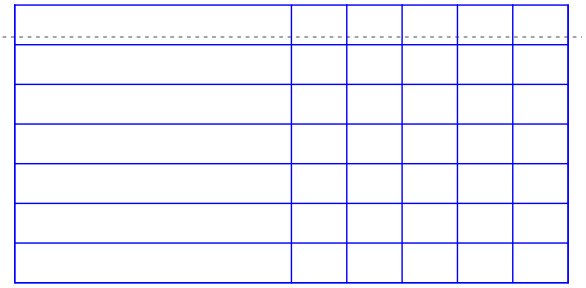


Figure 3

Use **Trim > Trim Object > Length / Radius to Object** you can shorten the middle four vertical lines to the Construction Aid line. Identify the upper part of each line and then click on the Construction Aid line. Repeat this for each of the four lines (Figure 4).

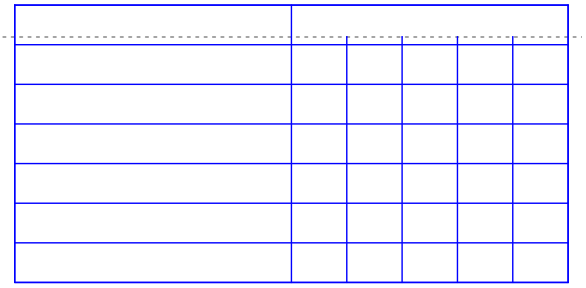


Figure 4

The basic structure of the table has now been set out. Next, put a line of text into the upper left cell. Use **Lettering > Text > Standard** and enter `Line Type` . The text should be left-aligned and 5 mm high (in the example, the Arial TrueType font is used). Close the dialog by clicking on OK and position the text the desired distance from the left edge. To align the text vertically, choose **Shape > Align Objects > Centered, Frame Vertical**. Identify the text and (with “Corner/ Endpoint” and “Midpoint” snapping modes active) click on the top right and bottom right corners of the box surrounding the text (see Figure 5)

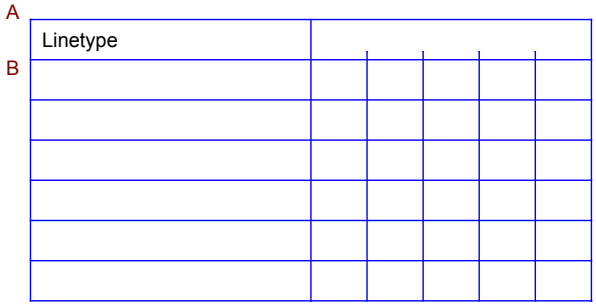


Figure 5

This text can be used as a template and be copied into all the relevant cells using **Construction > Multiple Copy > Gradual** Identify the existing piece of text and choose the upper left corner of the field as the reference point. The start point also lies at the top left corner (A) and the finishing point lies at the field's lower right corner (B). Choose six linear steps and click on OK (Figure 6).

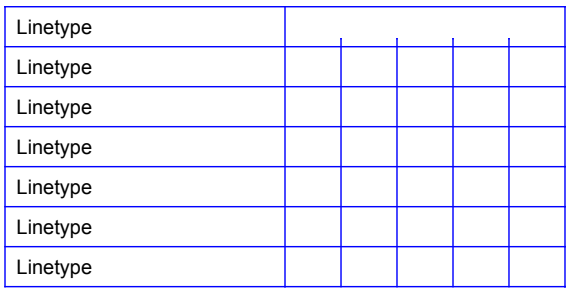


Figure 6

Use **Lettering > Text > Standard** and enter `Line Width / Line Groups` . Set the text alignment to "Centered", the text size to 4 mm and the text style to bold italic. Position the text in the space meant for it and use **Shape > Align Objects > Centered, Frame Both** to center it in the frame. Use the intersection of the Construction Aid line with the right hand vertical line as the lower right corner of the frame (Figure 7).

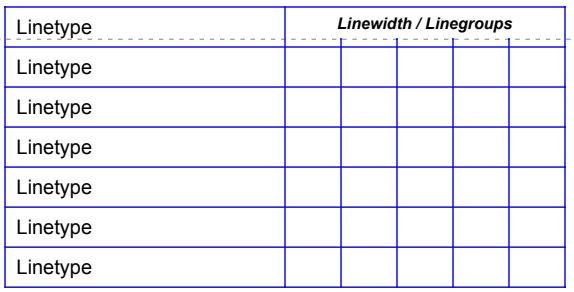
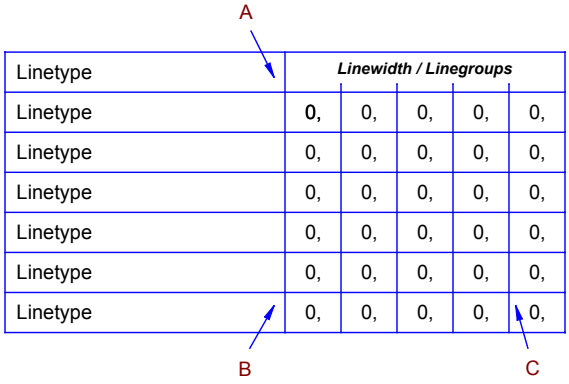


Figure 7

Use the same method to place the figure `0,` centered in the top left number field. It should be 5 mm high and use the standard font. Copy this text into all the other number fields. Choose **Construction > Multiple Copy > Array** and identify the text object. The reference point is the upper left of the field (A). This point is also the first corner point. Corner point 2 is at the top left corner of the bottom left number field (B) and corner point 3 at the upper left corner point of the bottom right number field (C). In the first dialog box enter `5` for the number of linear segments (six copies) and in the second dialog box enter `4` (five copies) for the number of horizontal segments. See Figure 8.



Linetype	*Linewidth / Linegroups*				
Linetype	**0,**	0,	0,	0,	0,
Linetype	0,	0,	0,	0,	0,
Linetype	0,	0,	0,	0,	0,
Linetype	0,	0,	0,	0,	0,
Linetype	0,	0,	0,	0,	0,
Linetype	0,	0,	0,	0,	0,

Figure 8

You can now use **Shape > Edit Text** to alter each piece of text to the correct value. Identify and alter each piece of text separately (see Figure 9).

Linetype	*Linewidth / Linegroups*				
Solid line wide	0,25	0,35	0,5	0,7	1,0
Dash-dot line wide	0,25	0,35	0,5	0,7	1,0
Dash line	0,18	0,25	0,35	0,5	0,7
Solid line narrow	0,13	0,18	0,25	0,35	0,5
Dash-dot line narrow	0,13	0,18	0,25	0,35	0,5
Freehandline	0,13	0,18	0,25	0,35	0,5

Figure 9

Alter the properties of the `Line Types` piece of text. Do this with **Shape > Edit Properties**. After identifying the text, click on the “Special” button and alter the text style to “Bold Italic”.

Finally, the tables appearance is improved by adding differing line widths and shading. Choose the pen “0.5 mm\Solid Line Wide” and (with “Corner/Endpoint” snapping mode active) draw the outer frame. Change back to the pen “0.25 mm\Solid Line Narrow” and use **Draw > Line > Equidistant** to draw a thin line 0.7 mm outside the last-drawn rectangle. To do this, identify the rectangle and click outside the surface. The corners should not be rounded (Figure 10).

Linetype	***Linewidth / Linegroups***				
Solid line wide	0,25	0,35	0,5	0,7	1,0
Dash-dot line wide	0,25	0,35	0,5	0,7	1,0
Dash line	0,18	0,25	0,35	0,5	0,7
Solid line narrow	0,13	0,18	0,25	0,35	0,5
Dash-dot line narrow	0,13	0,18	0,25	0,35	0,5
Freehandline	0,13	0,18	0,25	0,35	0,5

Figure 10

The dividing line in the middle of the table can be given the properties of the pen “0.5 mm\Solid Line Wide” using **Shape > Edit Properties**. To do this the relevant entry from the list in the dialog is chosen and enable the “Fix” check box.

To draw attention to the 0.5 and 0.7 line groups, a gray grid is placed behind them. Choose the pen “0.5 mm\Solid Line Wide” and draw a rectangle around the area to be shaded gray. Use the snapping function. Using **Shape > Edit Properties** , set the rectangle’s filling style to “Filling & Outline”, click on the “Color” field and choose gray (or any other color) for the background (Figure 11).

Linetype	***Linewidth / Linegroups***				
Solid line wide	0,25	0,35			1,0
Dash-dot line wide	0,25	0,35			1,0
Dash line	0,18	0,25			0,7
Solid line narrow	0,13	0,18			0,5
Dash-dot line narrow	0,13	0,18			0,5
Freehandline	0,13	0,18			0,5

Figure 11

Because the rectangle was drawn last, it is on top of the other elements in the table. Use **Shape > Change Order > To Back** to place the rectangle in the background (Figure 12).

Linetype	*Linewidth / Linegroups*				
Solid line wide	0,25	0,35	0,5	0,7	1,0
Dash-dot line wide	0,25	0,35	0,5	0,7	1,0
Dash line	0,18	0,25	0,35	0,5	0,7
Solid line narrow	0,13	0,18	0,25	0,35	0,5
Dash-dot line narrow	0,13	0,18	0,25	0,35	0,5
Freehandline	0,13	0,18	0,25	0,35	0,5

Figure 12

Export

Clipboard

There are various ways to make this graphic available for use in other programs. The usual way is to copy it to Windows' Clipboard using **Edit > Copy**. It can now be pasted into any program which uses vector graphics.

EMF

You can copy objects to an enhanced metafile using **File > Export > Enhanced Metafile**. The current output properties are used. Most 32bit Windows applications which work with vector data can read EMFs.

To import an enhanced metafile into Malz++Kassner CAD6, use the **File > Import > Enhanced Metafile** command.

If you have the choice to use either EMF or WMF, you should always prefer EMF files. They use 32bit coordinates, complex surface descriptions including Bézier curves, nested outlines and further high-level structures that result in high-resolution, device-independent data.

WMF

Alternatively, you can copy objects to a windows metafile using **File > Export > Windows Metafile**. The current output properties are used. Most 16bit Windows applications which work with vector data can read WMFs.

To import a windows metafile into Malz++Kassner CAD6, use the **File > Import > Windows Metafile** command.

BMP

A further possibility is export to a Windows Bitmap file. These can be read by many programs which handle bitmap graphics. Choose **File > Export > Windows Bitmap** to export to a bitmap.

To import a windows bitmap into Malz++Kassner CAD6, use the **File > Import > Windows Bitmap** command.

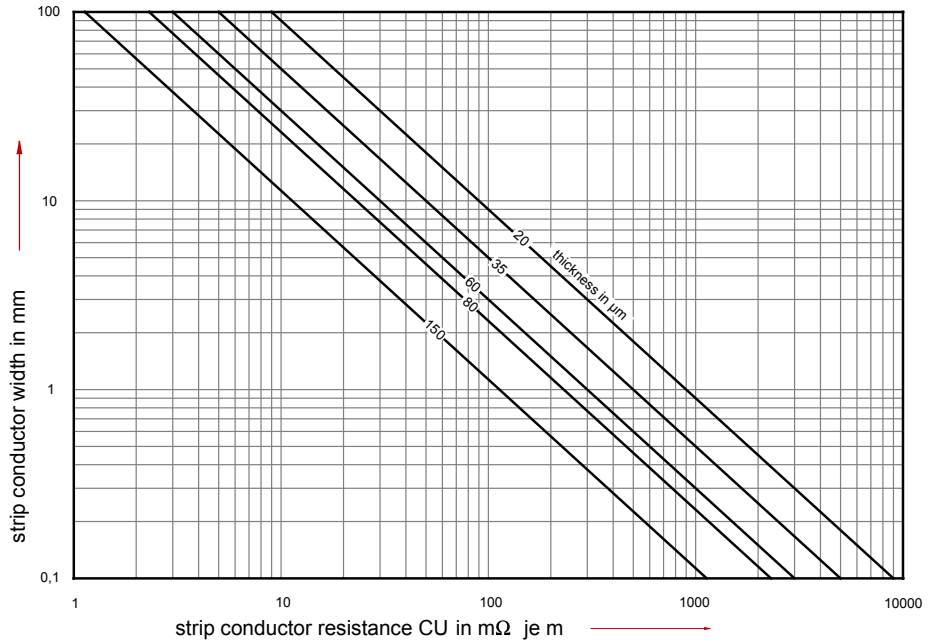
Settings

It is also possible to export parts of the drawing such as Construction Aid lines, markings and grids. This can be useful, for example, in technical documentation. Many of the graphics used in this tutorial were one with Malz++Kassner CAD6 and exported using suitable extra output parameters (markings and Construction Aid lines). The output parameters are set in a dialog called with **View > Drawing Settings > Output**. These settings can also be altered at the last minute by clicking on the “Options” button in the **File > Print Drawing** dialog.

In order to export parts of the current drawings to a Malz++Kassner CAD6 file, choose the **File > Export > Malz++Kassner CAD6 (*.MKD)** command.

If the program to which you wish to export Malz++Kassner CAD6 graphics cannot handle TrueType fonts, enable the “TrueType As Surface” check box in the output parameters dialog. This setting converts all characters into surfaces before output to the Clipboard, a WMF or the printer. This procedure can also be useful if TrueType fonts do not print correctly on your printer.

Diagrams



This chapter uses some commands that are only available in Malz++Kassner CAD6 Pro, CAD6 Studio, and/or CAD6 Industry!

Aim

Malz++Kassner CAD6 makes it possible to draw complicated technical drawings for various technical purposes. In this example a double logarithmic resistance diagram showing Ohmic conductor resistance against conductor width

Settings

`Tutorial.mkd`, with the page set to landscape orientation.

Set the boundaries of the diagram. Using the pen “0.25 mm\Solid Line Narrow” or a self-defined thin pen, draw the left and bottom boundaries of the diagram with **Draw > Line > Horizontal** and **Draw > Line > Vertical** Leave enough space to add further text. You can resize the diagram at a later stage, but getting it right now will save work later. So that the diagram looks tidy, use “Corner/Endpoint” snapping mode to make sure that the vertical lines do not extend below the intersections (Figure 1).

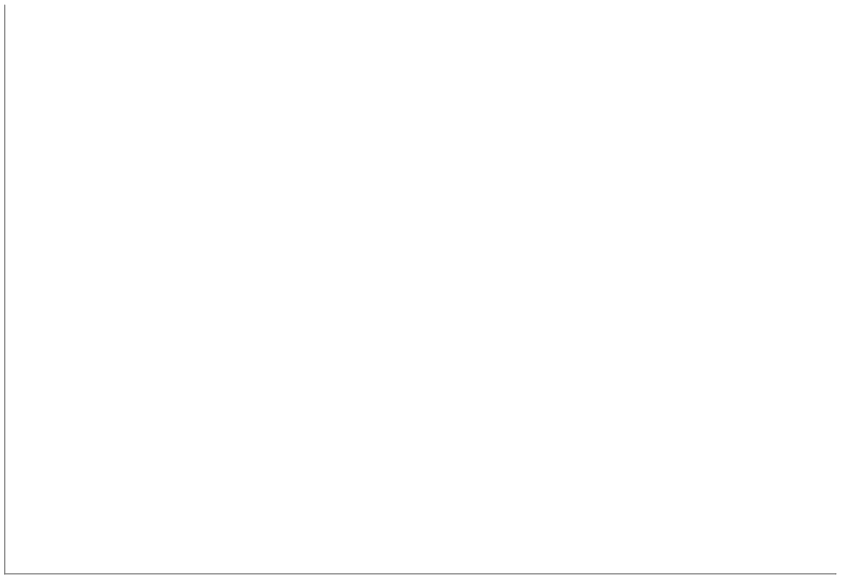


Figure 1

The diagram is divided into four decades in the X-direction and 3 decades in the Y-direction. To draw the grid lines use the command **Construction > Multiple Copy > Length**. Begin with the X-divisions. Identify the vertical line and (with “Intersection” snapping mode on) choose the intersection of both lines as the reference point. This is also the start point; the end point is the end of the horizontal line. Click on the Logarithmic radio button and enter 4 as the number of decades. Close the dialog by clicking on OK (Figure 2).

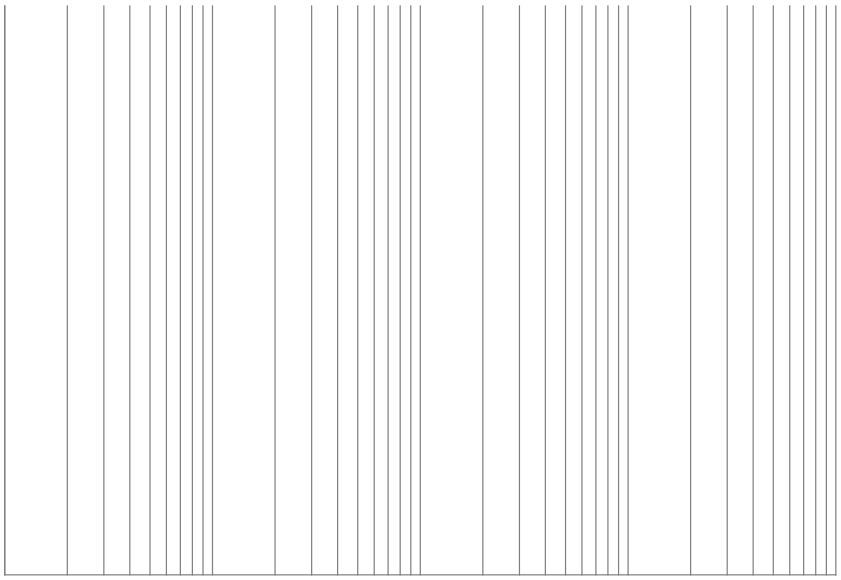


Figure 2

Proceed in a similar way to make the multiple copy of the vertical line. The start point is the same, but the end point is the top of the vertical line and there are three decades (Figure 3).

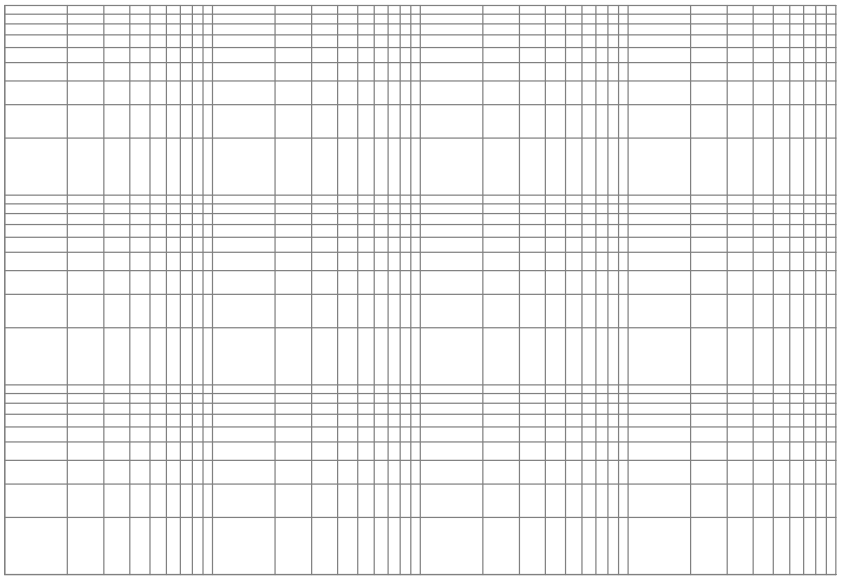


Figure 3

You have now created a piece of logarithmic graph paper as the grid for the diagram.

Next, label the axes at the appropriate points with appropriate values. In the example, 4 mm high text is used. The X axis labeling can also be placed rotated by 90°. To do this enter the rotation angle in the **Dimension > Text > Standard** To align the labeling with a line, draw a vertical and a horizontal Construction Aid line parallel to the relevant axis. You can place the label text on these lines making use of “Edge” snapping mode (Figure 4)

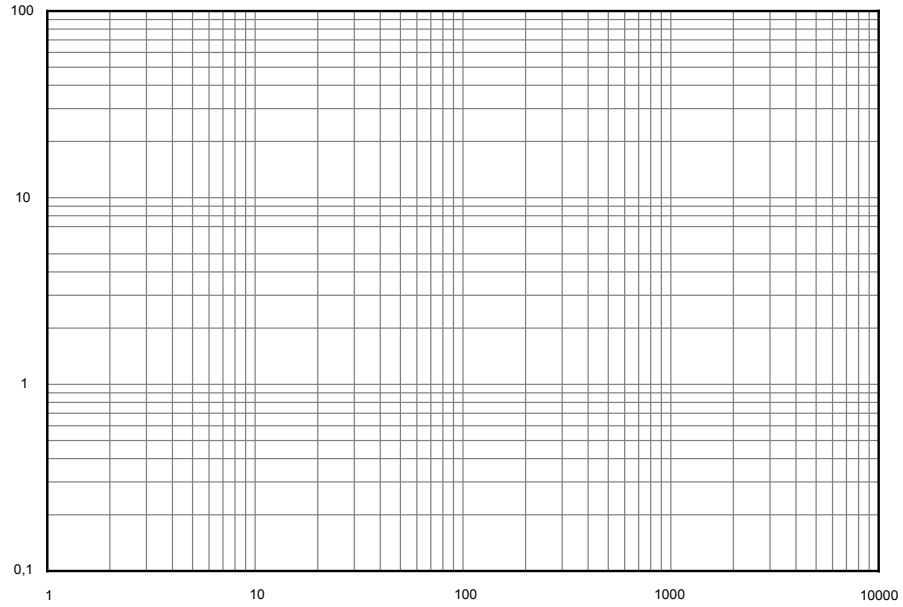


Figure 4

Now you can begin to draw the graph lines using the pen “0.5 mm\Solid Line Wide”. This can be done with the help of bases and Bézier curves or Splines, or, as here, with a straight line through defined points. Begin with the line (20 µm) through the points (9000/0.1) and (30/30). These points are already present as intersections in the diagram. Turn “Intersection” snapping mode on and click on the relevant intersections. To extend the line to the edge of the grid, use **Trim > Trim Object > Length / Radius to Object**, identify the upper part of the last-drawn line and click on the uppermost horizontal line as the reference line (Figure 5).

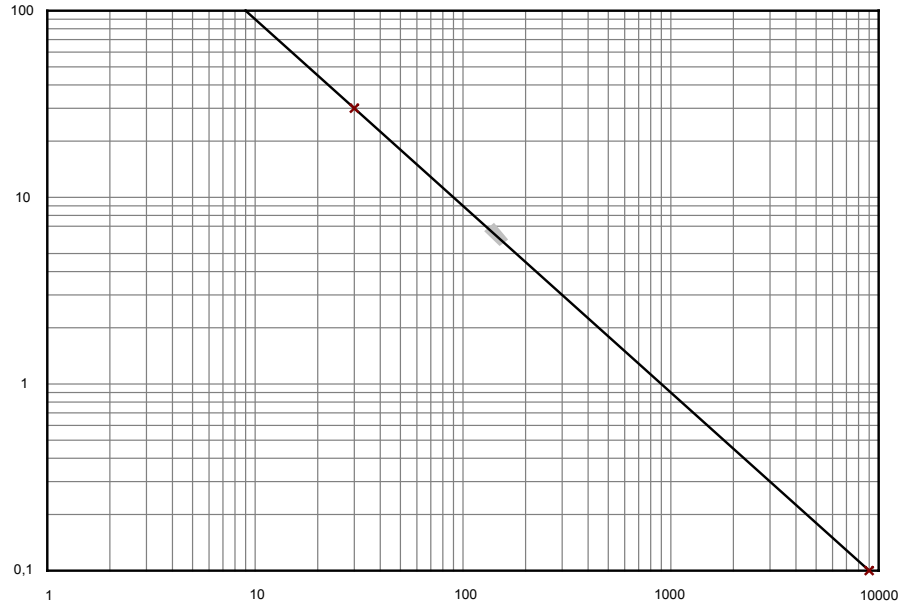


Figure 5

Because all the graphs in the example or parallel lines, the line has to be duplicated running through the new reference points. Activate the duplicate function by pressing F7 and use **Shape > Move Object > Perpendicular**. to move the line. Identify the graph line and choose any vertical gridline as the reference line. The point chosen as the reference point is very important. A point should be chosen which brings the new graph line onto an intersection or a previously placed mark. This point now has to be moved to the new target position. For the first copy, choose the lower corner point and move it to (5000/0.1). Do the same for the other graph lines. You can take suitable reference points from the example diagram (Figure 6).

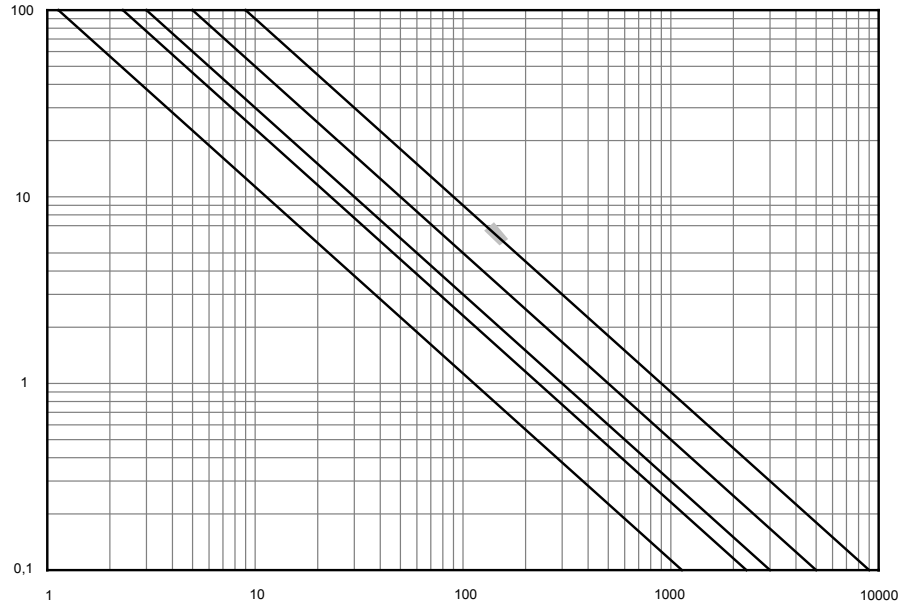


Figure 6

So that the lines can be labeled with text running at the correct angle, the line angle has to be measured. Use **Lettering > Dimension > Angle, Line - Line** to measure the angle between the graph line and the horizontal grid lines. The parameters are not important because after noting the angle, the dimension is immediately removed by using **Edit > Undo** (BACKSPACE).

Enter the angle, preceded by a minus sign (-) (the text is rotated clockwise, mathematically negative) into the **Lettering > Text > Standard** dialog. In the example 3 mm high text is used. Place the text up against the graph line in the diagram. It is not necessary to go through the whole process for each new piece of text - it is enough to restart the command using SHIFT+ESC (N) and enter a new text (Figure 7).

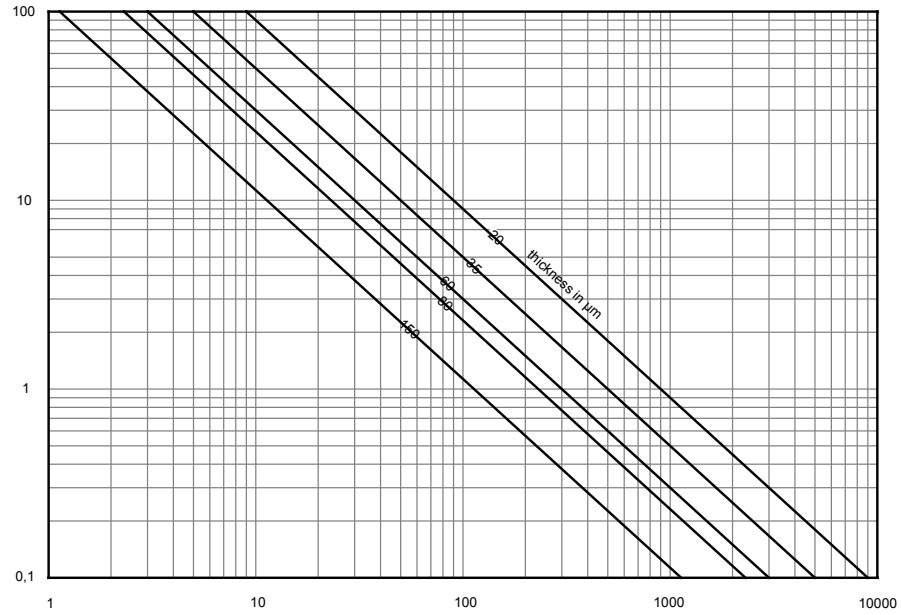


Figure 7

The texts are directly above the lines and are difficult to read. To make them easier to read, place an erasing surface behind each piece of text to hide the grid lines. Use **Draw > Polygon > Arbitrary** to draw a surface covering a piece of text, set its filling mode to "Eraser" using **Shape > Edit Properties** and then use **Shape > Change Order > To Front** to bring the text to the front (Figure 8).

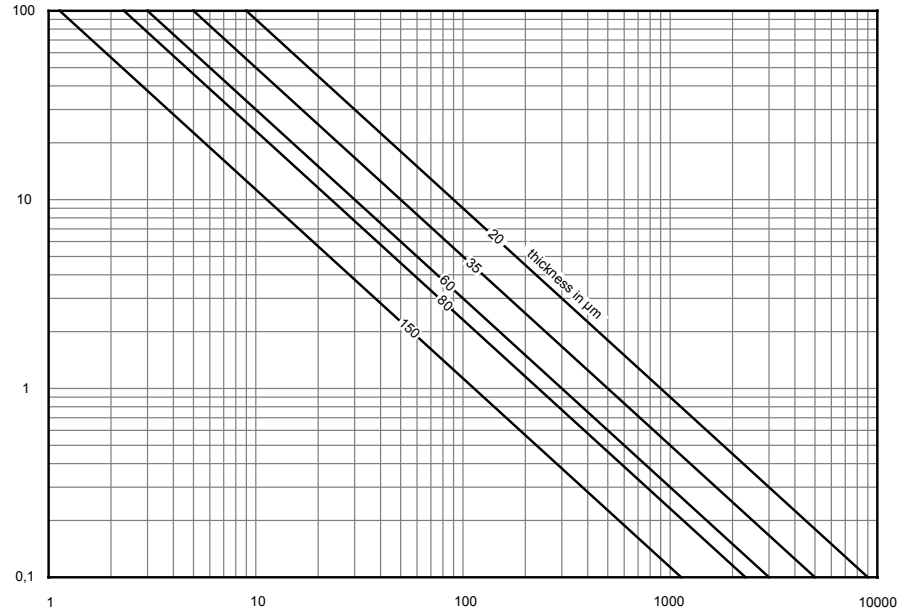


Figure 8

Finally, label the axes. Enter the text using **Lettering > Text > Standard** The Omega symbol is from a different font and is added separately. The arrows are straight dimension lines. To align them correctly place a Construction Aid line (**Construction > Construction Aid Endless Line > Horizontal**) in the middle of the X-axis label. Using **Lettering > Dimension Line > Straight** with "Edge" snapping mode active, the dimension arrow can be aligned exactly horizontally. Proceed in a similar way for the vertical dimension arrow (Figure 9).

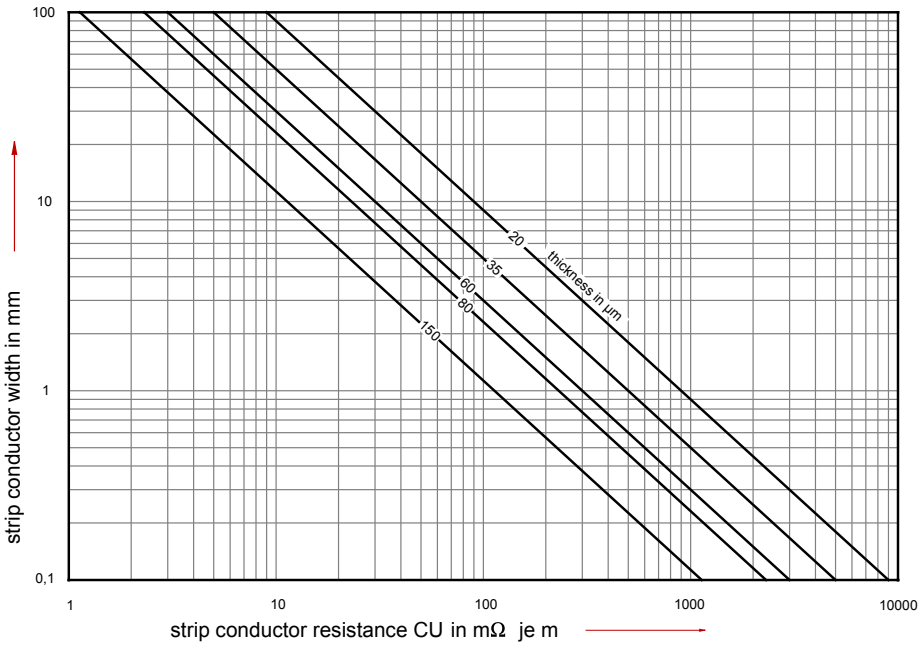


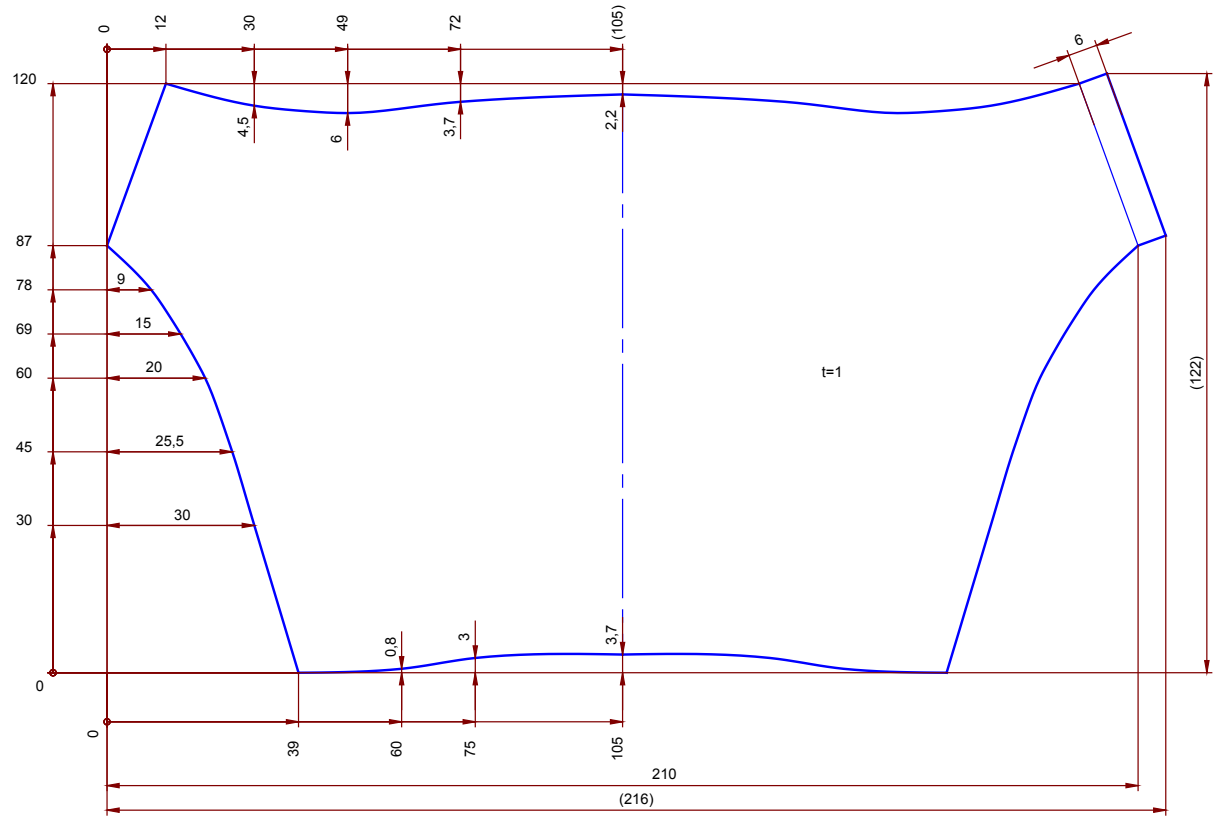
Figure 9

Alternative Forms of Diagrams

Linear or logarithmic diagrams can easily be created in the same way. During the multiple copy process, a linear division is used for one or both axes.

Non-linear graph lines, of which only coordinate points are known, are approximated using Splines or Bézier curves. This is explained in the following example (the Trouser Leg).

Trouser Leg



This chapter uses some commands that are only available in Malz++Kassner CAD6 Pro, CAD6 Studio, and/or CAD6 Industry!

Aim

This example demonstrates the use of Spline curves in technical drawing and further dimension measurement techniques.

Settings

`Tutorial.mkd`, with the page set to landscape orientation.

Basics

The most important parts of the design are three curves which determine the outline. The curves have defined fixed points and during drawing a Spline curve is passed through those points. As the drawing is virtually symmetrical, it is only necessary at this time to draw the left hand side, which can then be mirrored with the duplicate function active.
Begin the drawing by using **View > Coordinate Systems > Set Origin** to place the drawing origin in the page's bottom left corner. This is not absolutely necessary but it makes the representation easier, as some of the points would either lie on the edge of the page or would have to have an offset value added to their coordinates.

The points are set using markings, Choose **Construction > Markings**, press F8 and enter the first point's coordinates. Use the values from the drawing at the beginning of this chapter. A reference point measurement is chosen, and the values are relative to the origin. For your convenience, so-called redundant measurements, calculated from existing measurements, are given in brackets. If possible, the use of these measurements in drawings should be avoided.

For the lower curve, the following points have to be entered using markings:
(39; 0) (60; 0.8) (75; 3.0) (105;3.7).
See Figure 1.

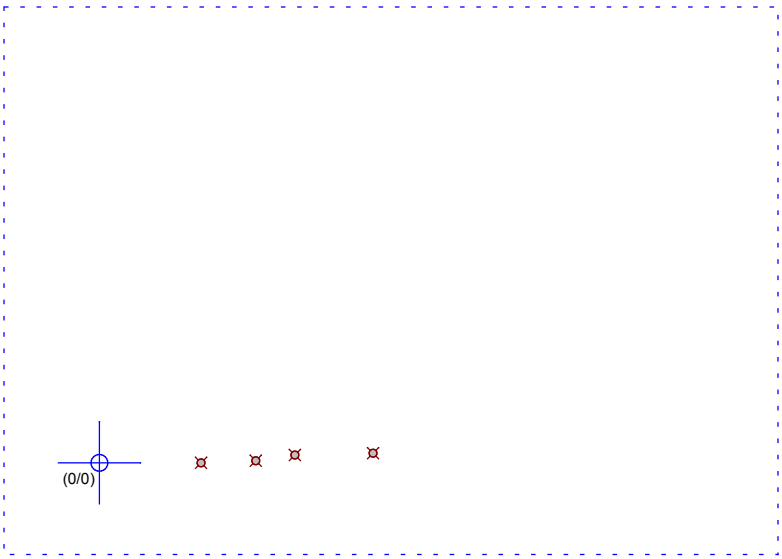


Figure 1

Proceed in the same way for all the points in the left half. To specify the position of the upper curve, use the formula `y=120-a` where a is the distance to the highest point of the curve (Figure 2).

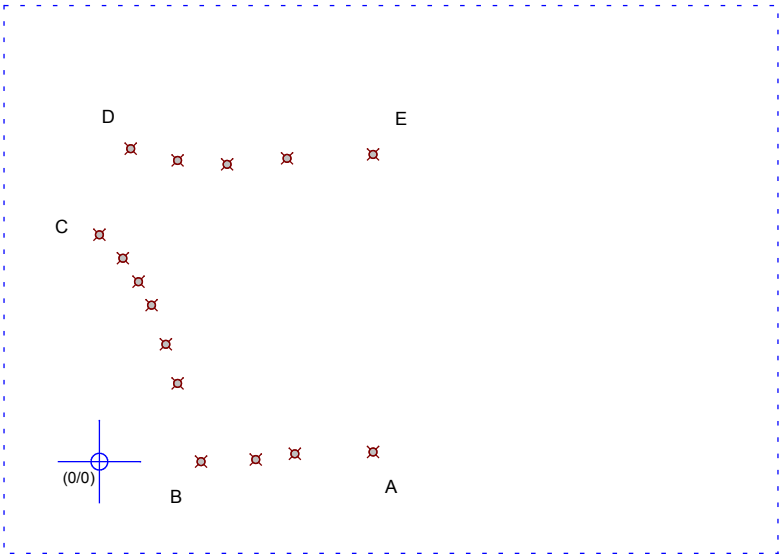


Figure 2

The outline of the left side of the trouser leg consists of three curves: AB, BC, DE and a line CD. Note that the curves AB and BC are two separate objects. They must not be drawn as a single object as they form a corner at point B. Corners cannot be defined within a Spline curve. The command must be ended at point B and then restarted at the same point.

Choose the pen “0.5 mm\Solid Line Wide” to draw the rest of the objects. Activate “Markings” snapping mode and join the points with a Spline curve (**Draw > Spline**). To finish drawing a Spline, click the right mouse button, and if the Workflow Manager is active and appears, choose **Finish Command**. Join points C and D using **Draw > Line > Standard** with “Markings” snapping mode active (Figure 3).

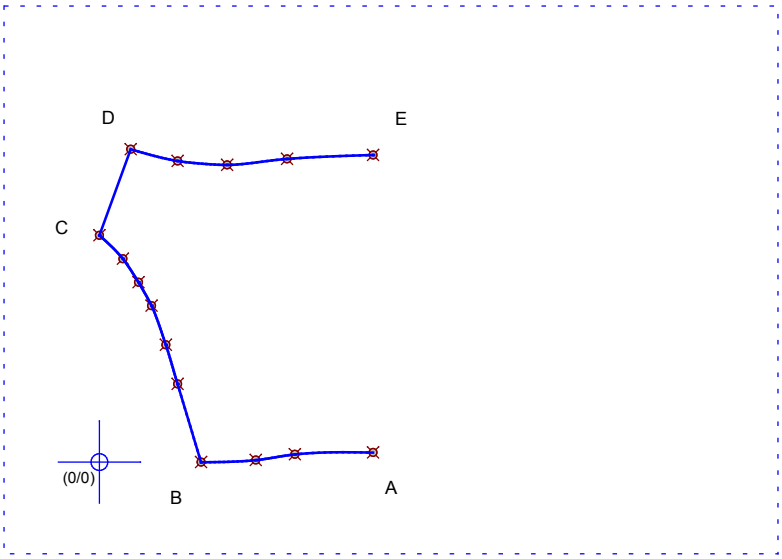


Figure 3

Next, the symmetry axis is drawn between points A and E. You can now mirror and duplicate the outline using **Shape > Reflect Objects > Line** Hold the CTRL key down whilst identifying the four objects. Release it and then press it again. This activates the duplicate function. Now identify the symmetry axis to reflect and duplicate all the selected objects (Figure 4).

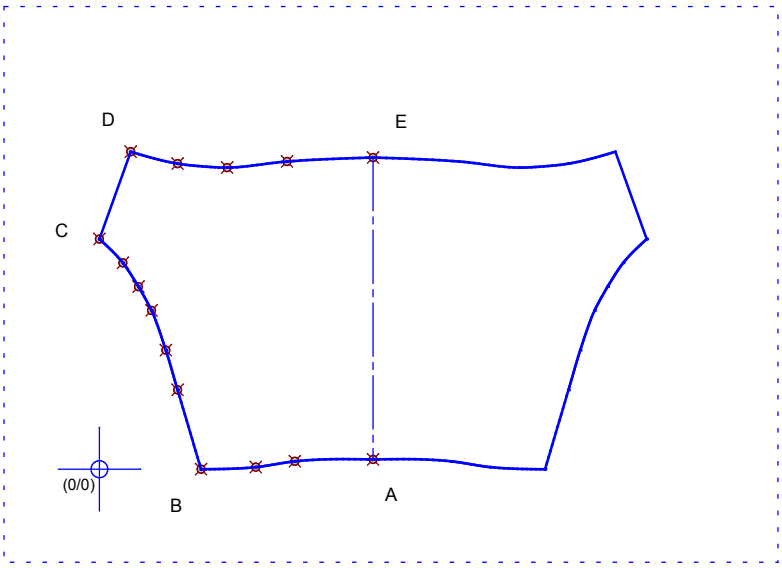


Figure 4

To draw the tongue on the right hand side, move the existing line by 6 mm parallel to itself with the duplicate function active. Choose **Shape > Move Objects > Perpendicular** and

identify the line on the right. Choose the line itself as the reference line. Press F7 to turn on the duplicate function and then press F8 (ENTER) and enter 6 mm. Notice the arrow direction in the dialog. This shows the positive measurement direction - for example, if the arrow is pointing down and to the left, then you have to enter -6 mm as the distance to avoid copying in the negative coordinate direction.

Choose the pen “0.5 mm\Solid Line Wide” and join the endpoints of both lines using **Draw > Line > Standard**. To indicate a fold line, use **Shape > Edit Properties** to change the inner line from pen “0.5 mm\Solid Line Wide” to pen “0.25 mm\Solid Line Narrow”. This completes the outline of the trouser leg (Figure 5).

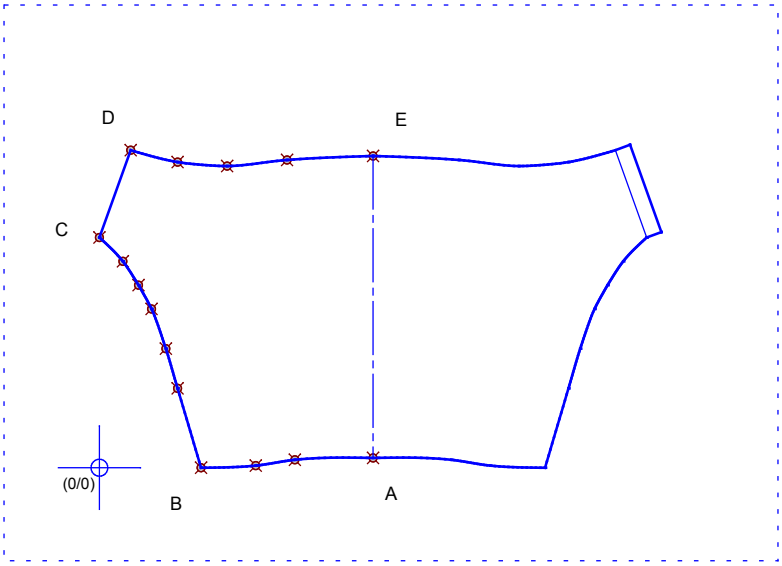


Figure 5

It is best to add the dimensions to the left hand side first. Begin by using **Lettering > Dimension Parameters** to set the text size to 3 mm so that the dimension text will fit within the shorter measurements. To help with aligning the dimension texts use **Construction > Construction Aid Endless Line > Vertical** at the X-position -13 mm. In addition use **Construction > Construction Aid Endless Line > Horizontal** to draw horizontal Construction Aid lines throw all the markings on the left hand Spline curves. The dimension lines are placed using the intersections of these Construction Aid lines as reference points (Figure 6).

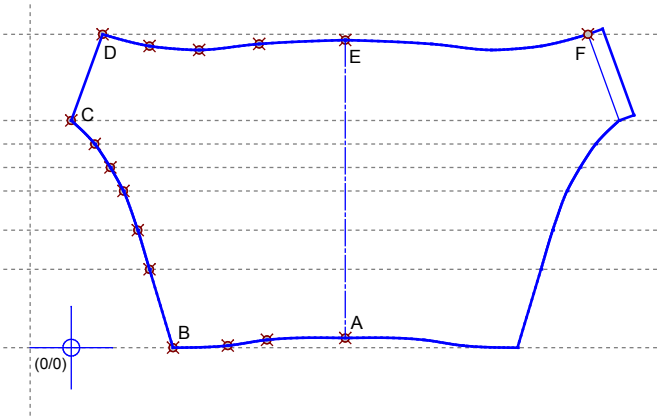


Figure 6

Start by measuring the distances between the curve points of the left-hand curve. Because there are no objects to use as references, use **Lettering > Dimension > Distance, Point**.

In the dimension parameter dialog (accessible with SHIFT+ESC), specify an open circle as the start symbol for the dimension line, but leave the end symbol as a filled arrow. In addition, click on the icon for vertical dimension line alignment and enter 5 mm as the distance. Deactivate the “Center” and “Close” check boxes (to enable hand-centering of the dimension text) and activate the “Rotate” check box.

The start point for *each* of the vertical dimensions is point B. This is also the reference point for the distance. The end point is the corresponding marking on the curve. Pull all the dimension lines out by the same distance so that they form a continuous line. This should be to the left of point C so that the upper part of the outline is not cut. To position the dimension text, snap to the to the Construction Aid intersection which lies at the same height as the marking at the end point of the extension line. Enter the dimension text’s rotation angle (0°) into the dialog after pressing F8 (ENTER).

The last dimension, 120 mm, is an exception. The end point for this measurement is placed on the reflection of point D (F), to create a longer extension line. It is not necessary to draw a further line later on (Figure 7)

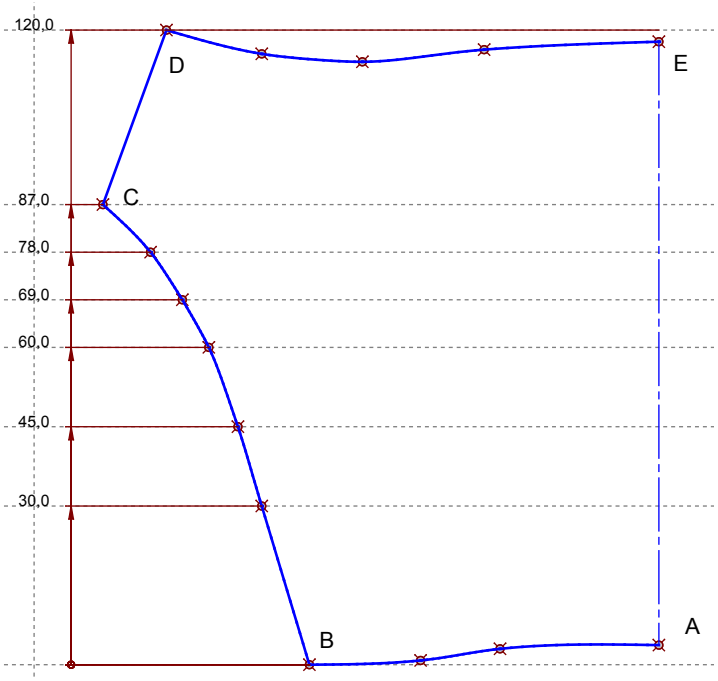


Figure 7

Proceed in a similar manner for the horizontal sections on the top and bottom of the outline. Use point C as the starting point (see Figure 8).

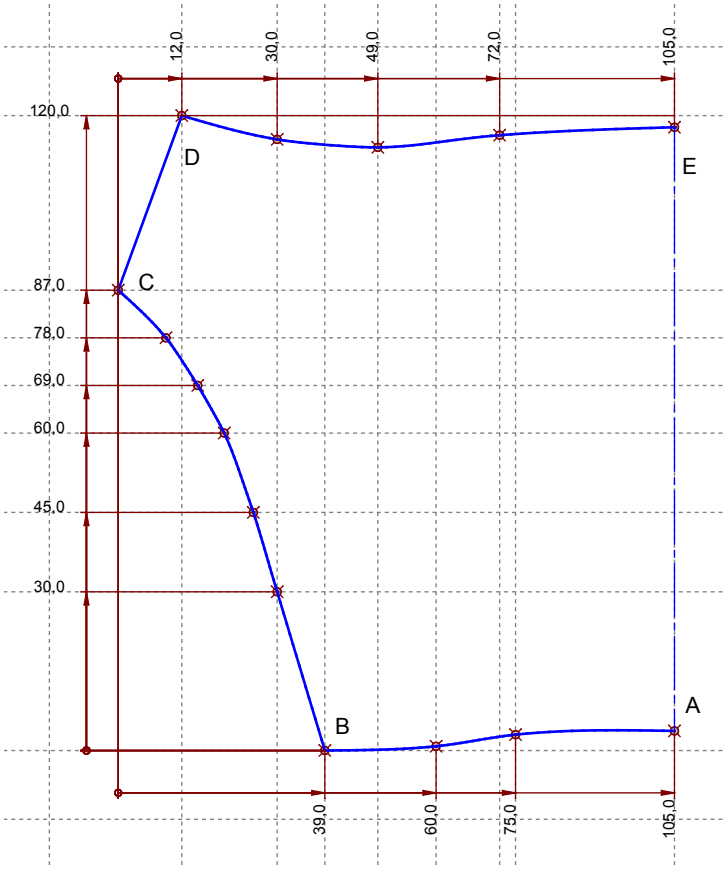


Figure 8

Next, the missing overall dimensions are added. When drawing the vertical height measurement of 122 mm it is important to place the starting point on point B as the intersections of extension lines for measurements drawn later will be placed there. Remember to set the start symbol back to an arrow in the dimension parameters dialog. The check box “Close” can be reactivated, and the check box for rotating the dimension text can be deactivated.

To have the right separation distances available for all the dimensions, reduce the dimension line distance to 5 mm in the dialog. Use **Lettering > Dimension > Distance, Object - Object**. to apply the 6 mm dimension to the tongue. In this case, it is not necessary to use snapping as you only have to identify the objects (Figure 9).

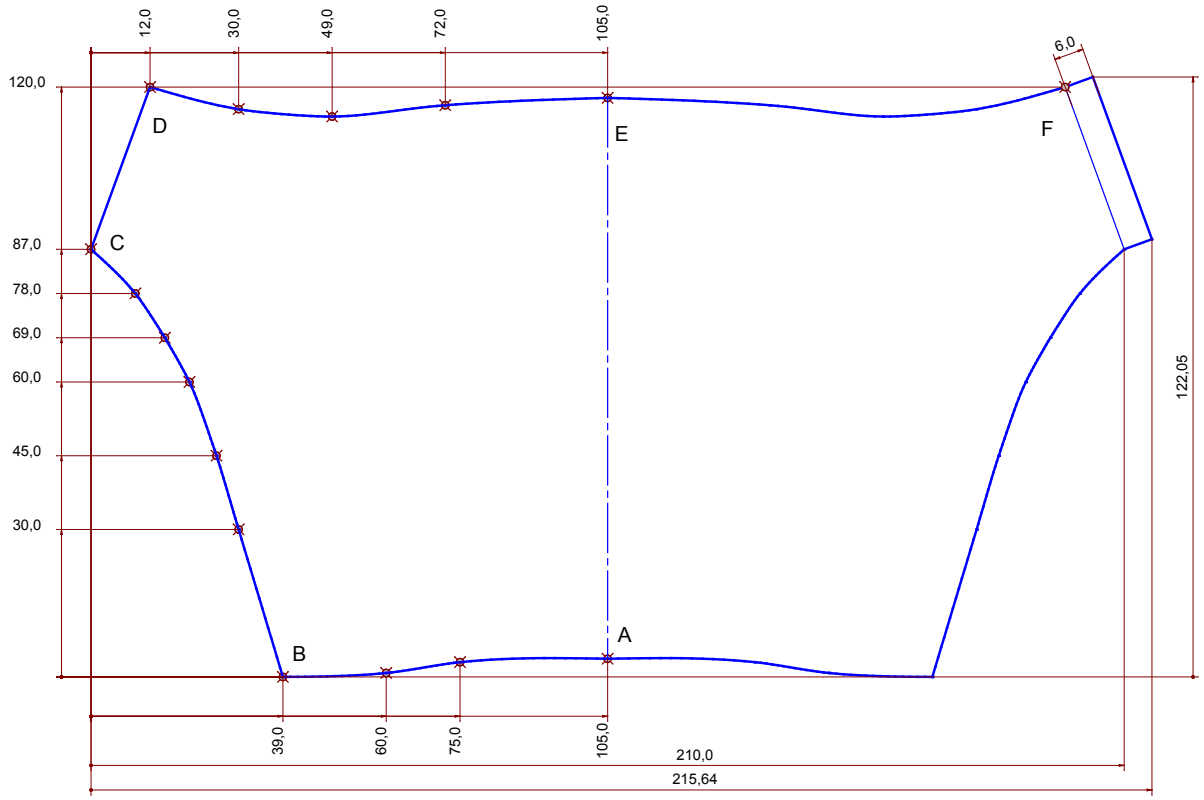


Figure 9

To add the distances from the curve points to an imaginary rectangle enclosing the outline, use the command **Lettering > Dimension > Length, Point**. again. Choose the filled arrowhead as both starting and end points of the dimension line and deactivate the “Center” check box. Turn snapping on, and click on the marking and the corresponding intersection of the section measurement with the adjacent extension line (see Figure 10).

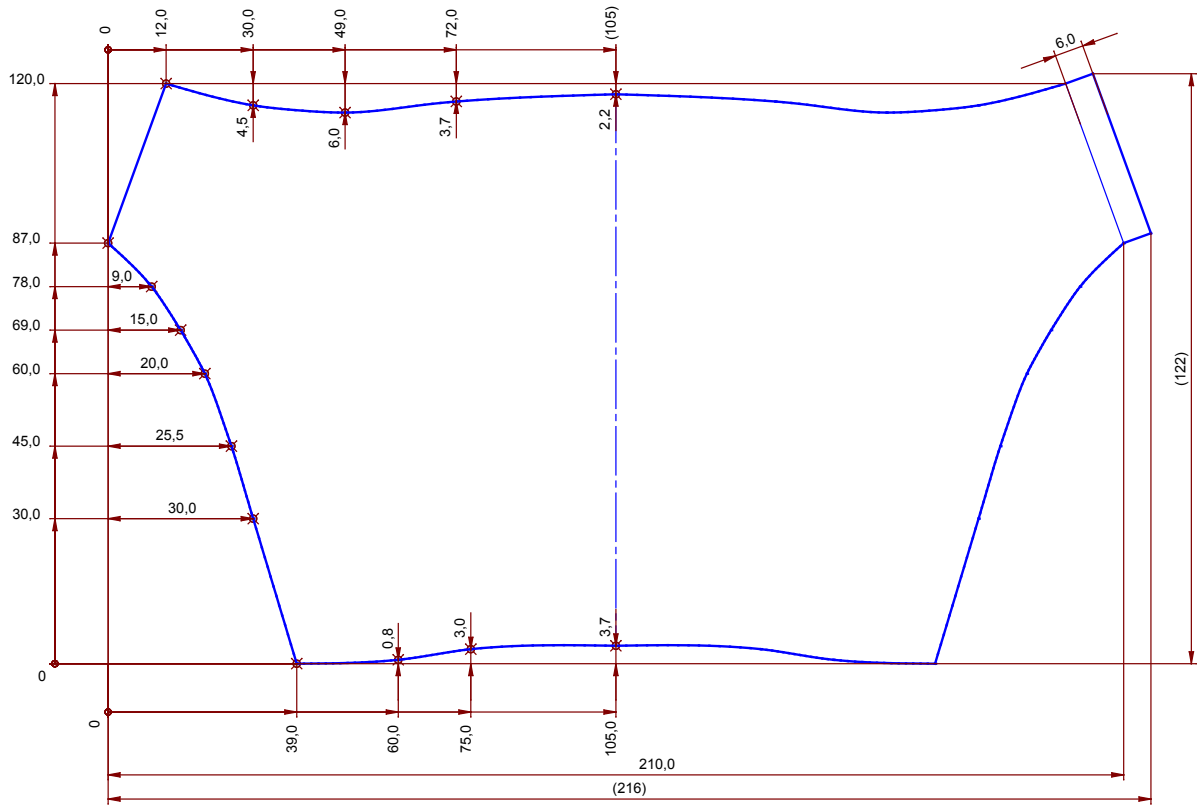


Figure 10

Finally, use **Shape > Edit Properties** to alter the general parameters for the redundant measurements to no decimal places. This can be carried out in one operation, by clicking on each of the affected measurements in turn holding down the CTRL key. In the Edit properties dialog, click on the “Special” button to get the general properties. Each individually identified dimension can then be placed in brackets to mark it as an informational dimension. Do this by using **Shape > Edit Text**. Use **Lettering > Text > Standard** to place the text `t=1` (a note about the thickness of the metal to use) in the middle of the right hand side. (Figure 11).

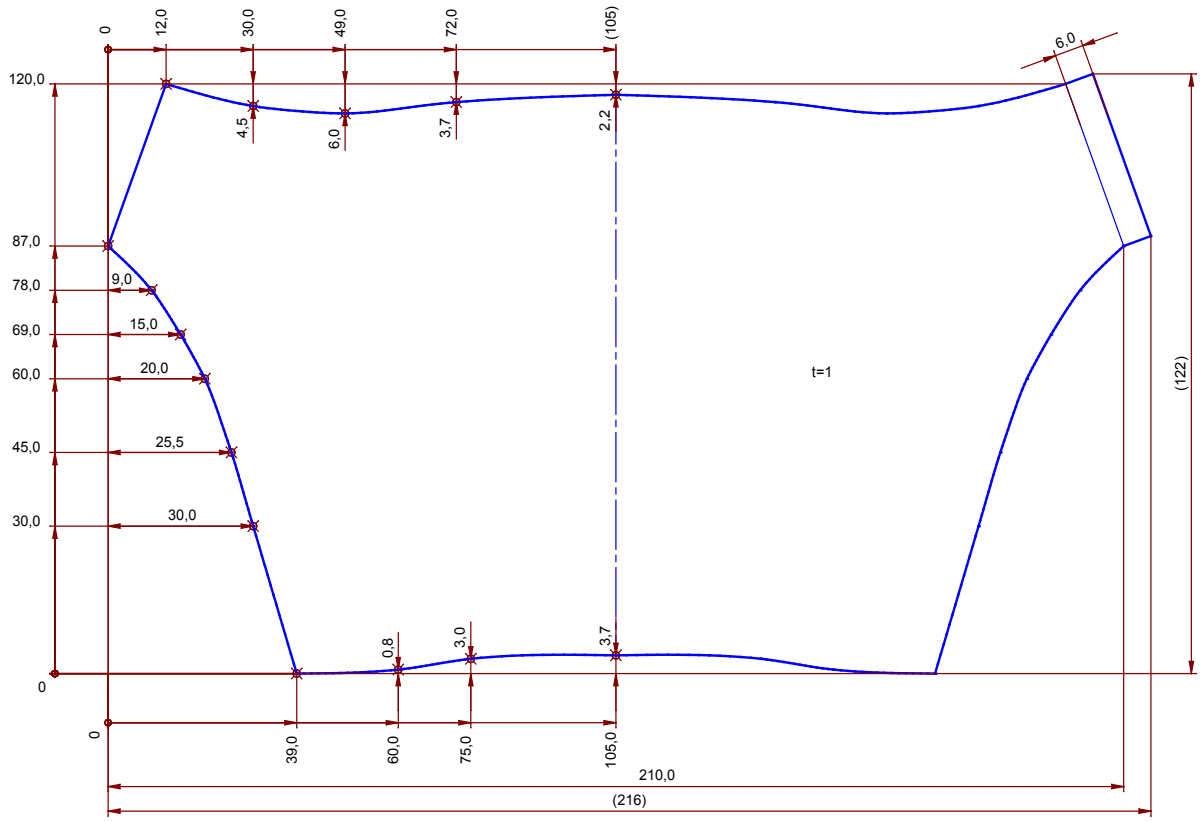


Figure 11

Lastly the drawing can be centered on the page by choosing **Shape > Align Objects > Centered, Page Both** and pressing F10, then it can be saved using **File > Save Drawing as**.

Libraries

Concepts

Groups

In drawings, the same elements are often used repeatedly. The element can be either a single object or several objects placed together in a group. The easiest way of managing objects which belong together in a drawing is to use a g*roup*. Groups are made by using **Shape > Group > Group Objects**. Objects which have been included in a group can only be altered or moved together. The advantage is that you do not have to identify each individual object before altering the group's properties. The program recognizes automatically that it is dealing with a group and reacts accordingly.

If you make a number of objects into a group, Malz++Kassner CAD6 puts the objects into memory and substitutes a reference to these objects in the drawing. A reference like this is known as an i*nstance*. The original objects no longer exist on the page. Only an image of them is present. The advantage of this procedure becomes clear when you make a copy of the objects. Each further copy of the group consists of an instance, that is, a reference to the group held in memory. Each instance requires only the same small amount of memory. This makes it possible to manage several large (memory intensive) groups in a drawing using the least possible amount of memory. Instances are saved, along with the group's object information, in drawing files.

When groups are manipulated, this structure is usually not disturbed. Like other objects, each instance has its own set of properties. When an instance is manipulated, only the on-screen representation, not the original objects, are changed. Malz++Kassner CAD6 only applies the alterations to the original objects when an instance is resolved and the original objects are pasted into its place from memory. If the group contains further instances, the changed properties are applied to the relevant instances and fixed into the resolved objects.

Example

Use a simple example to see how a group behaves. Draw an element made up of three basic objects and make them into a group using **Shape > Group > Group Objects**. To create a group, identify each object to be included in the group in turn. Manipulate this group with commands from the **Shape** menu. The title bar of the **Shape > Edit Properties** dialog shows information about the type of object. You can use **Shape > Group > Ungroup** to break the group up into its individual objects.

Internal Blocks

It is awkward to manage large numbers of repetitive groups, and so Malz++Kassner CAD6 makes it possible to name groups and include them in a list. Each named group can be called as an independent element from the list. These named groups are called i*nternal blocks*. Internal blocks are read into a library using **Library > Block > Create (Insertion Point)** or **Library > Block > Create (Frame)** As for the command **Shape > Group > Group Objects** the relevant objects must be identified. To read in an internal block, the library "*Internal Blocks" must be chosen in the dialog, that is, the path must begin with "*Internal Blocks". This is the description for a block which will be saved with the drawing. You can enter a name with up to 64 characters for the block in the "Block:" field. You can then specify using a check box whether the objects which make up the newly saved internal block should remain in the drawing or be replaced by an instance.

To place a block from the list into the drawing, choose **Library > Block > Insert** and choose an internal block from the list. Make sure that the "*Internal Blocks" library has been selected. Before clicking on OK, you can enter a custom rotation angle and/or scaling factor for the block. The chosen block now "hangs" at the cursor. If the block was read in via frame, then the insertion point will always be in one of the insertion frame's corners. During insertion, the insertion point always lies at the center of the cursor. You should bear this in mind while reading in the block and use a distinctive point as the insertion point. Use snapping when you insert blocks into the drawing.

External Blocks

If a block is stored in an external library, a separate file, then it is known as an *external block*. Before an external block can be created, a library must be opened or created using **Library > Library Management**. and clicking on either "New" or "Open" and then entering either a new or existing library name. Click on OK to either open the existing library or create a new library. You can now read a new external block into the library in much the same way as reading in an internal block. You can make use of the commands **Library > Block > Create (Insertion Point)** or **Library > Block > Create (Frame)**. You must set the correct path to the library in the dialog which appears.

To more clearly arrange internal and external blocks, a hierarchical folder structure, similar to that used for file directories, can be created. To create a folder, use the **Library > Block > Edit** command.

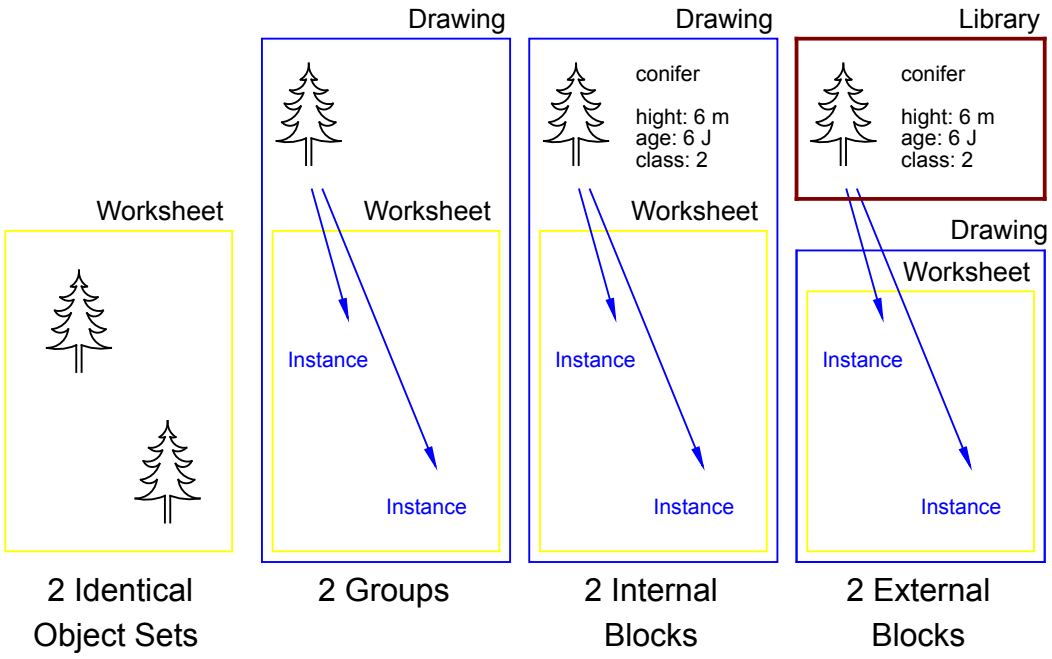


Figure: Groups, internal and external blocks

It is only possible to change the properties of instances in a drawing. The original, in memory, remains unchanged. Malz++Kassner CAD6 does however make it possible to change the properties of blocks directly. Using **Library > Block > Edit** you can alter the original (not an instance of the block). Such an alteration will be visible immediately, because any instances in the drawing are based on the altered original in memory. If individual instances in a drawing may not change you can either convert external to internal blocks (**Library > Convert External Blocks to Internal**), resolve blocks (**Library > Block > Resolve Block Instances**) or make a copy of the relevant library block (**Library > Block > Edit**).

Comparison

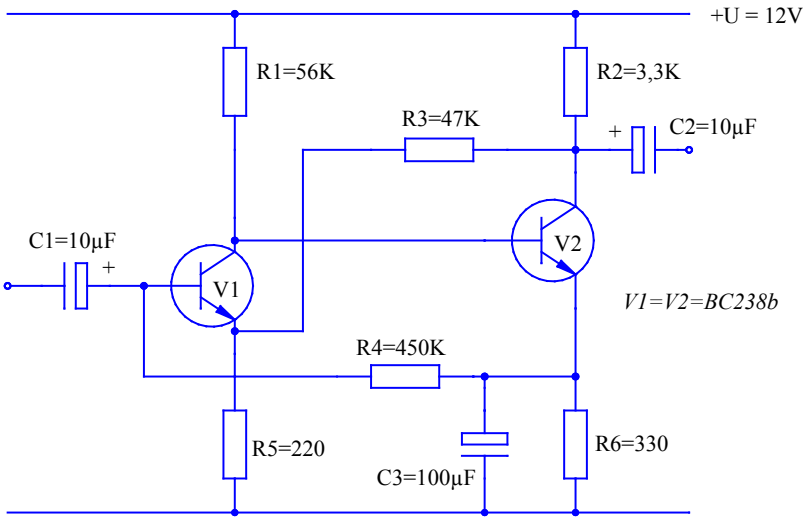
Groups are the simplest, but most inconvenient way to manage objects which belong together. They are used to protect objects temporarily from alteration or to create simple structures during drawing.

Internal blocks should be used in drawings when it is important to work independently of libraries. Whenever external blocks are used in drawings, it is important to make the corresponding libraries available as well. Internal blocks make it easy to manage elements and prevent the drawing from being changed by accidentally altering a library. Basically, an internal block should be used if an object is only used in one drawing.

External blocks should be used whenever the same block is used in several drawings. Using external blocks ensures consistency, because if an external block is altered then that change is automatically applied to all Instances of that external block in all drawings. After altering an external block, drawings only have to be printed again to get the correct, altered drawings. In addition, the use of external blocks referring to libraries reduces the size of the saved drawing file.

The external blocks in all drawings change, without any form of checking, control, or notification, if the libraries which contain the block definitions are altered. Because of this, the use of external blocks should be thought about carefully!

Circuit Diagram



two-stage amplifier

Aim

This example illustrates the practical use of libraries in a drawing. You will practice creating a new library, merging and altering elements in the library. In addition, methods for working with simple background grids are introduced. This example is drawn from the world of electronics, but the same techniques can be used in similar ways in other areas of work.

Settings

Tutorial.mkd

Basics

A circuit diagram consists largely of repetitive elements (yellow parts of figure 1) which, when they have been draw once, can be used again in other drawings.

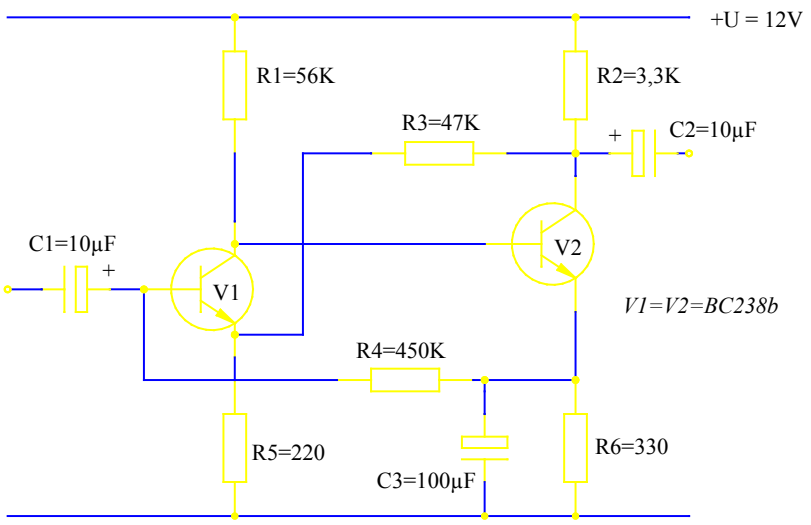


Figure 1

In order to make these elements as widely usable as possible, they should be drawn in accordance with recognized standards. For example, there is a DIN Standard for the background grid behind the drawing elements. In this example the size of the background grid should be set to M=2 mm. To create universally usable elements on this grid junctions should always lie at 2M (i.e. 4 mm).

Preparations

Before beginning drawing, turn on the display grid using the F3 key or the button in the Panel. Select 4 mm grid size so that only every other grid point, that is the points to which the crosshair snaps are displayed. The grid size can be set in the dialog called with **Coordinates > Coordinate Systems > Edit** - "Display Grid" or by clicking the right mouse button on the corresponding button in the Panel. Choosing 2M as the grid spacing has the advantage that screen redraws are quicker and you can also see exactly which points the connections between components lie on. In addition, turn on the position grid by pressing F4 or clicking on the button in the Panel. The position grid size should be set to 2 mm. The position grid settings can be specified in a dialog called with **View > Coordinate Systems > Edit** - "Position Grid" or by clicking the right mouse button on the corresponding button in the Panel.

Begin by creating a new library. Choose **Library > Library Management** to call a dialog. Click on the “New” button and enter `Electro.mkl` into the “Library Name” field. After closing the dialog by clicking on “OK”, enter `Electronics Library` as a description for the library. This title will be used later in the list of libraries. Next, find the new library in the list of libraries. It will still be prefixed by an ‘N’ to remind you that this is a new library whose contents are determined after leaving the dialog. Leave the dialog by clicking on “OK”. The new library has been created, but it is still empty.

Choose a suitable zoom level (one which gives a good view of each component) to work on the components. For example, try a zoom factor of 4.

Our simple circuit diagram contains four basic library elements: resistors, capacitors, transistors and contact points. Start with the simplest component.

Resistor

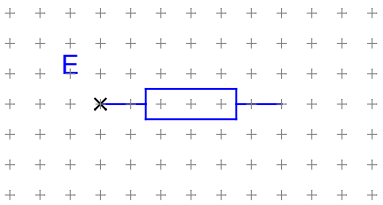


Figure 2

The resistor consists of a rectangle and two short lines. Take the dimensions from the diagram. You do not need to use snapping as the position grid is active. The crosshair can only be moved in 2 mm increments. Every grid point displayed on screen is 4 mm from the next grid point because the display grid is set to 4 mm.

Once you have drawn the resistor with three objects, read it in to the library using **Library > Block > Generate (Insertion Point)**. To do this, identify all the objects and click on the contact point at E as the insertion point. Choose the new “Electronics Library” in the Path section of the dialog which appears. Enter `Resistor` as the component name and leave the dialog by clicking on “OK”. You have now added the first component to the library.

Capacitor

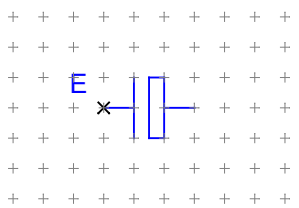


Figure 3

The capacitor is also relatively easily drawn from three lines and a rectangle. Be careful to ensure that all the connectors lie on grid points. Use the same procedure to read the capacitor into the library as for the resistor. The name of that block should be `Capacitor` .

Transistor

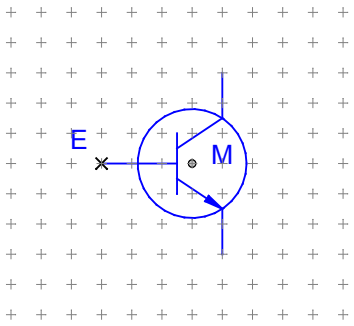


Figure 4

Begin by drawing the connections. The line with the arrow is drawn using Lettering > Dimension Line > Straight Make sure that the "Not rotated" setting in the dimension line parameter dialog (accessible via SHIFT+SCR) is enabled or the arrow will be pointing in the wrong direction. Finally, draw the surrounding circle using **Draw > Circle > Standard**. The center of the circle is the display grid point marked with an M (Figure 4).

The radius of the circle is the point of the arrow just drawn. Choose connector E as the insertion point and read the component into the Library using the same procedure as for the resistor. To avoid confusion with other transistors which might be added to the Library later, give this transistor the name `NPN-Transistor` .

Contact points

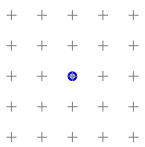


Figure 5

The contact point is a circle with a 0.5 mm radius circle. Draw it on top of a grid point using Draw > Circle > Standard and use the direct entry method (F8) to enter the radius 0.5 mm. Read this object into the Library with the name `Open Contact Point` .

Next alter the circle's properties with **Shape > Edit Properties** to solid fill (Figure 6).

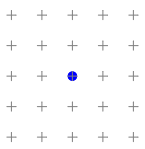


Figure 6

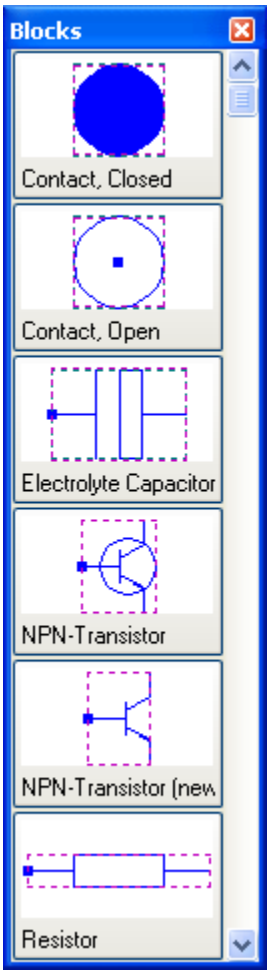
Save this object as `Closed Contact Point` . The insertion point should be in the center of the circle.

Circuit

All the required components are now stored in the library. Before you draw further, make sure the blocks are saved. Do this with **Library > Library Management**. Select the file to be saved, "Electronics Library", from the dialog and click on the "Save" button. An 'S' now appears in front of the library name. Leave the dialog by clicking on OK. The library is saved when leaving the dialog.

Use the command **File > New Drawing** to start drawing the circuit diagram. Activate the position and display grids with the same values used when drawing the components (2 mm position grid and 4 mm display grid). Use **View > Pages > Edit** to choose A4 landscape format and choose a zoom factor of 2 with **View > Zoom > Factor**.

Even though not strictly needed as the diagram is not very complicated, the use of the block list window is introduced at this point. Choose **Window > Block List on/off**. The block list window will now appear on the right of your screen. Frequently used blocks can be stored here. They can be activated by clicking on them in the block list window in a similar way to choosing a command from the toolbox window.



To place a block into the block list window, click the right mouse button on one of the buttons in the block list window. This calls a dialog which is the same as the one called with **Library > Block > Insert**. Choose the library which you wish to use. Then click on the block name and choose a rotation angle and enlargement factor. As the components in the library just created are already the right size, only the rotation angle needs altering. Begin by choosing the transistor; leave the dialog by clicking on OK without altering the rotation angle. The NPN transistor should now appear in one of the block list window buttons. Proceed in the same way for the other components. Add the resistor again, but rotated by 90°. If the blocks' rotation angles are not displayed, choose **Edit > Options > Block List** and enable the "Rotation" checkbox.

Start by positioning the more complicated blocks. In this diagram this means the transistors. First, position transistor V1. Click on the transistor icon in the block list window. The component “hangs” from the crosshair. The insertion point lies at the center of the crosshair. If the grid is active, this guarantees that the block will be inserted with its end on a grid point. Insert the element by clicking the left mouse button.

Next choose the vertical version of the resistor and place it twice on the page aligned with the upper and lower connectors of the transistor (resistors R1 and R5). You only need to position them approximately as they can easily be moved using **Shape > Move Objects > Standard** If the grid is on, the movement is restricted so that the components always jump to the grid points and there is no need to use the snapping functions. Use the same method to position capacitor C1.

Insert transistor V2 at its approximate position. The insertion point should be a little higher than the upper connector of V1 and enough space should be left between the two transistors for R4 and C3. The resistors R2 and R6 are positioned in a similar way to R1 and R5. Next, place the horizontal resistors R3 and R4 into the drawing. A suitable block is already available in the block list window.

The capacitors C3 and C2 are rotated by 90° and 180° respectively and therefore are not in the block list window. Choose **Library > Block > Insert** and choose the “Electronic Library” as the path. Click on the “Capacitor” block and choose 90° as the rotation angle. Close the dialog with “OK”. The block is now available. Insert it into the diagram and carry out the same procedure for the second capacitor (180° rotation) (Figure 7).

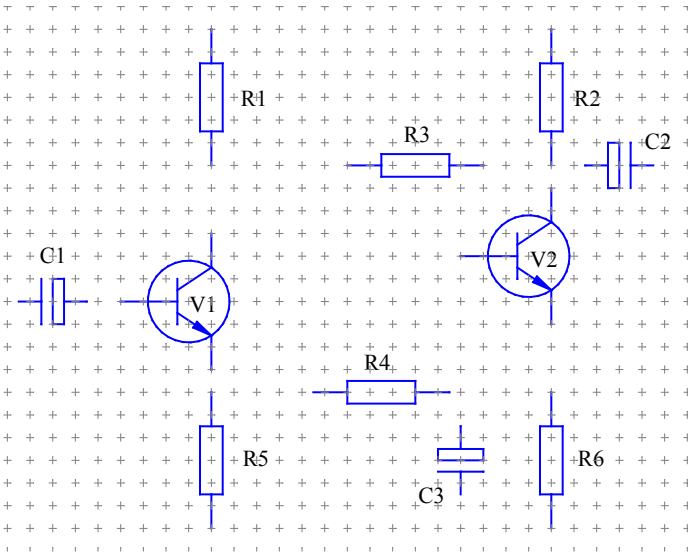


Figure 7

Use **Draw > Line > Polyline** to draw connecting lines with 90° corners, for example between V1 and R3, on the grid. The other, short, connection lines can be added using **Draw > Line > Standard**.

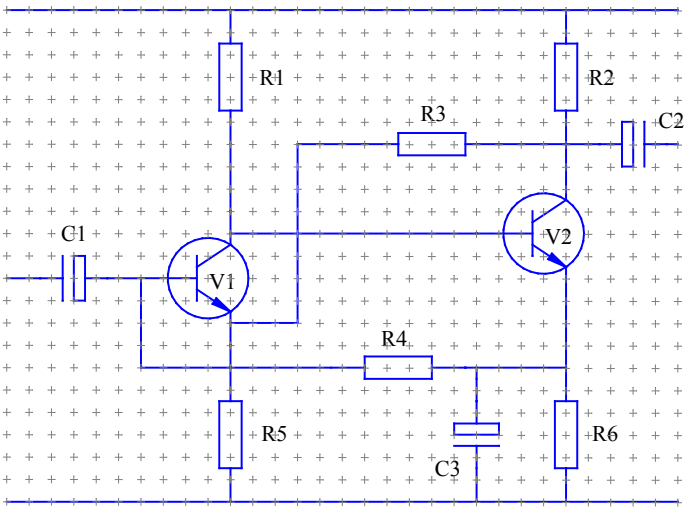
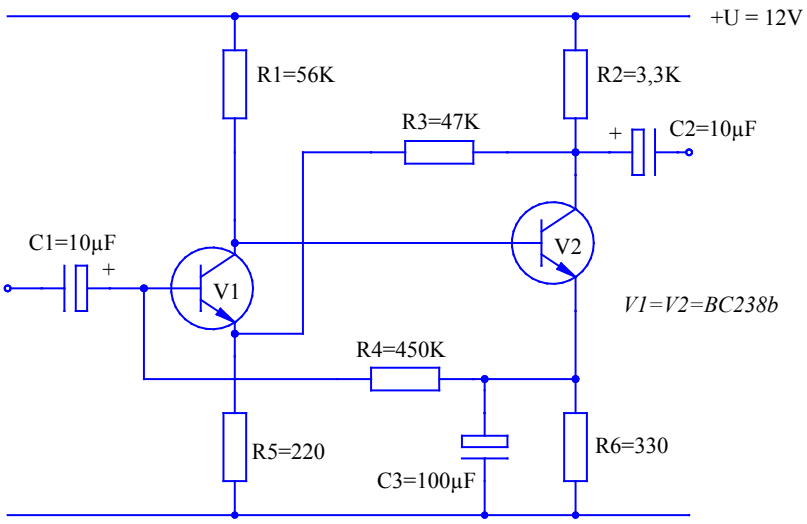


Figure 8

Remember that a polyline is actually a curve on which some operations can not be carried out. To manipulate the polyline, it must first be converted to a series of individual lines with **Trim > Trim Objects > Resolve Completely**, which can then be manipulated easily.

Next, place the block ""Closed contact point" on the junctions. Place an '"Open contact point" on the free ends of the capacitors C1 and C2. Finally, use **Lettering > Text > Standard**. to label the components. In the example, 4 mm high Times New Roman is used.

To enter a special character into the text, you can either hold down the ALT key and type the ANSI code for the relevant character on the numeric keypad and then release the ALT key, or start the Windows Character Map accessory, choose the character required, copy it to the Clipboard and then paste it into the Malz++Kassner CAD6 text field by using CTRL+V (or SHIFT+INS). This is how you add the µ symbol to the labels for the capacitors.



two-stage amplifier

Figure 9

To conclude this example, the working of libraries with respect to changes is explained again. Before carrying out the further exercises, save the drawing with the name `Circuit.mkd` using **File > Save Drawing as**.

Updating

One advantage when libraries are used is the easy maintenance of drawings. If the block is changed after the creation of a library, then all occurrences of that block in all drawings are updated as well. Test this by using the library which you created for the circuit diagram. Make a copy of the library with the name `Electronics Library Copy` by using the command **Library > Library Management** - "Save As". Later you will be able to copy the new library back to the original, undoing the changes. Before starting the exercises, start a new drawing by choosing **File > New Drawing**.

There are two main methods of placing new or altered blocks which will replace blocks in existing drawings into a library.

Method 1

The altered block is saved under a new, different name. This method makes it possible to only alter those blocks which you really want to change. It is the safest method and should therefore be used most of the time. Its disadvantage is that a relatively high amount of effort is required, because each block must be altered in each drawing.

In the example, if the standard for depicting a transistor changes, it would be necessary to make a corresponding alteration in the library. The new representation is different from the old one in that the circle around the block is no longer used. To change the representation, place the "Transistor" block on the page and use **Library > Block > Resolve Block Instances** to break it into objects. Remove the circle using **Shape > Delete Objects** (or the DELETE key). Read the altered block back into the "Electronic Library" using **Library > Block > Create (Insertion Point)**. If you did not have the grid active, activate "Corner/Endpoint" snapping mode to put the insertion point at the left-hand connector. Give the block the name `NPN Transistor (New)` to distinguish it from the old one. Save the altered version of the library by choosing **Library > Library Management** and choosing "Save" in the dialog.

Load your drawing `Circuit.mkd` . Using the command **Library > Replace Block** you can specify which block in the loaded drawing should be replaced by the new block. Choose the blocks by clicking on "Choose" and then selecting the blocks in the library dialog. All blocks with the old name will automatically be replaced by the new block. This operation can be restricted to permanently selected objects by activating the "Selection Only" check box.

Method 2

The second method is to directly overwrite the old block with the new one. This has the risk that blocks which you did not mean to alter will be altered. One advantage is that when the drawing is opened, it will already have been altered automatically.

The procedure is similar to the first method, except that the altered block is saved under the same name as the old one; it is not saved separately. After loading the drawing, you will see that the substitution has already been made.

Transforming

Remove the "Electronics Library" from memory by choosing **Library > Library Management** and clicking on the "Remove" button then leaving the dialog by clicking on "OK". The block name and library name for each block will be displayed on screen because the program cannot call on the library any more. Without the library, the circuit diagram cannot be displayed or printed properly, because the drawing only holds references to the original blocks in the library. After loading the library (with **Library > Library Management** - "Open") the blocks will be displayed again.

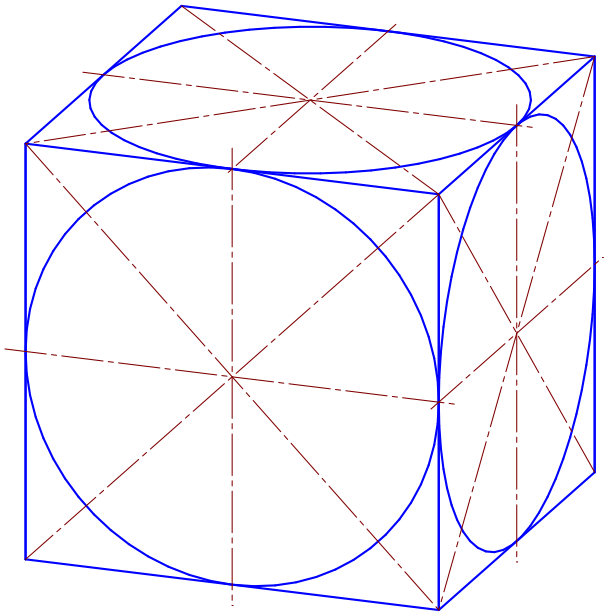
To make the drawing library-independent, either transform all the external to internal blocks with the command **Library > Transform External Block to Internal** or making the blocks into objects again using **Library > Block > Resolve Block Instances**.

Transforming external blocks to internal has the advantage that all the instances are kept intact and the blocks are saved with the drawing. Change the external blocks to internal and remove the library from memory. As expected, the drawing retains its original appearance. After saving the drawing, the blocks are contained, independent of the library, in the drawing.

Attributes

Each block can have unique properties (also known as *Attributes*). The block properties can be the same for all components or be different from component to component. Malz++Kassner CAD6 allows attributes to be attached to blocks and then output later in a parts list. Attributes which do not change for the same block are called *Global Attributes*. Individual properties, specified when a block is inserted into a drawing, are known as *Local Attributes*. Attributes should be set when a block is read in to a library.

Dimetric and Isometric Drawing



Aim

This example shows the main methods of working with dimetric and isometric views. You will learn the basic method for creating a 3-D view, using the example of a cube. We only explain dimetric projection because isometric projection uses the same methods.

Settings

`Tutorial.mkd`

To see the three additional views as well as the standard view, choose **Edit > Options > Windows** and choose the last-but-one icon from the top row. This gives you an extra three small windows arranged vertically at the right of the program window in which you can see different views. Click on “OK” to confirm your choice and close the dialog.

Basics

Dimetric and isometric projections are used to draw 3-D views of objects. Malz++Kassner CAD6 makes it possible to see each of the three views in the overview. This means that you can draw each side view in the same way as the two dimensional examples. However, circles appear as ellipses in isometric and dimetric drawing and must be drawn using **Draw > Ellipse > Rectangular** In a similar way, squares should be drawn as polygons using **Draw > Polygon > Parallelogram** The reason for this is that the program only makes the current view available as an aid to drawing. You are actually drawing in the dimetric view and have to use the commands which draw objects correctly in this view.

The Cube

Coordinate Systems

Each drawing window uses a different coordinate system. These systems must first be set up. Do this by choosing **View > Coordinate Systems > Edit**. Click on “New” to set up a new coordinate system. Give the first system the number 2 and the description `Dimetric 1, left view` . Create a further two systems, numbered 3 and 4, named `Dimetric 1, right view` and `Dimetric 1, plan view` . Next you have to apply the coordinate systems to the different windows. Click on a name and then on the “View” button. You can then choose the corresponding dimetric representation. The names of the coordinate systems are the same as the representations.

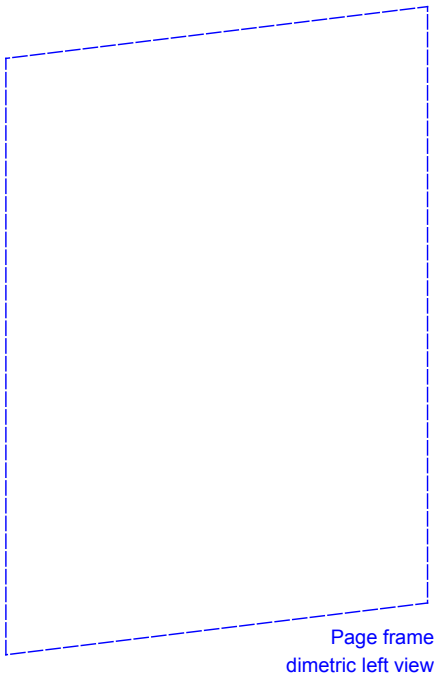


Figure 1

After you have generated all three new coordinate systems, choose coordinate system 1, “Drawing & Dimension” and leave the dialog. In order to apply the newly generated coordinate systems to the small windows, click on the upper left corner of the relevant window. The window number is displayed there in [square brackets]. The ruler display shows which window is active. After activating one of the small windows, you can apply a coordinate system to it by choosing **View > Coordinate Systems > Edit** and setting the coordinate system to the desired one in the dialog. Activate the dimetric views for windows 2-4.

Windows

Working with windows is relatively easy. You can easily choose the window you wish to work in. You can tell which window is active because the crosshair appears completely in that window, whereas in the others just a small cross showing its position appears. Even at

high resolution, windows 2-4 do not offer much drawing space and you can swap the contents of a small window into the main drawing window to make life easier. Do this by holding down the SHIFT key and pressing the number of the window whose contents you want to see in the main window. For example, SHIFT+3 will swap the contents of window 3 into the main window. The main window has the number 1. This makes it easy to put the view of your choice into the main window (Figure 2).

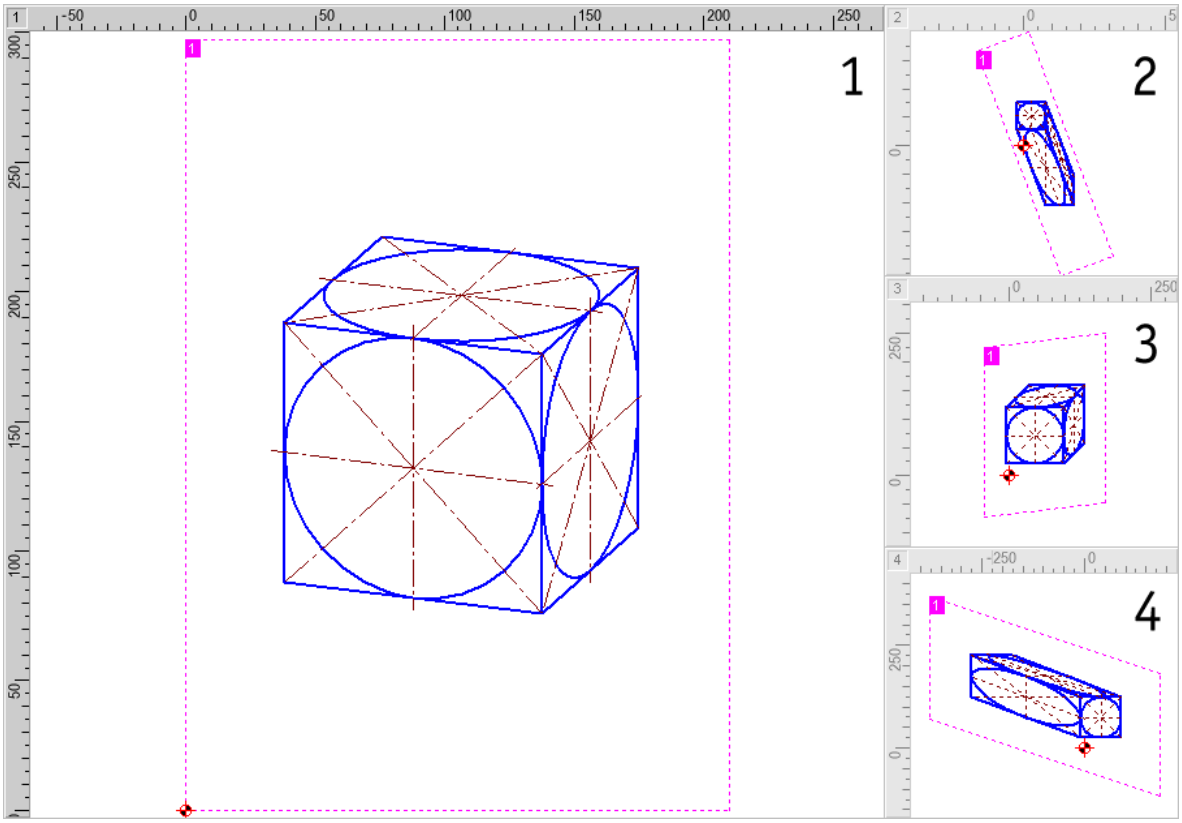


Figure 2

It is a good idea to always put the dimetric window back in its original place. This guarantees a better overview of the views. Otherwise, the left and right views (especially when working with isometric projection) can easily be swapped accidentally. If you have the left dimetric view in the main window and wish to change this with the right dimetric view, firstly put the left dimetric view back in window 2 (the uppermost of the three small windows) using SHIFT+2 and then bring the right dimetric view into the main window with SHIFT+3.

Grid

To make constructing the cube easier, turn on the position grid (size 5 mm) and the display grid (size 10 mm) for all three dimetric views. This is done most easily with the buttons in the panel. Activate the grids by left-clicking on the button and specify the grid size in the dialog called by right-clicking on the button.

The Drawing

Construct the dimetric left view in the main window first (check that you have the correct coordinate system in the dialog called by choosing **View > Coordinate Systems > Edit**.) Begin by drawing a square with sides 100 mm (that is, 10 grid points) long, using the pen "0.5 mm\Solid Line Wide". Use **Draw > Polygon > Parallelogram**. to do this. To have the same origin for the grid in the other views, set the zero-point to the upper right corner of the square. Do this with **View > Coordinate Systems > Set Origin**. Place a marking (U) on this point using **Construction > Markings** (Figure 3).

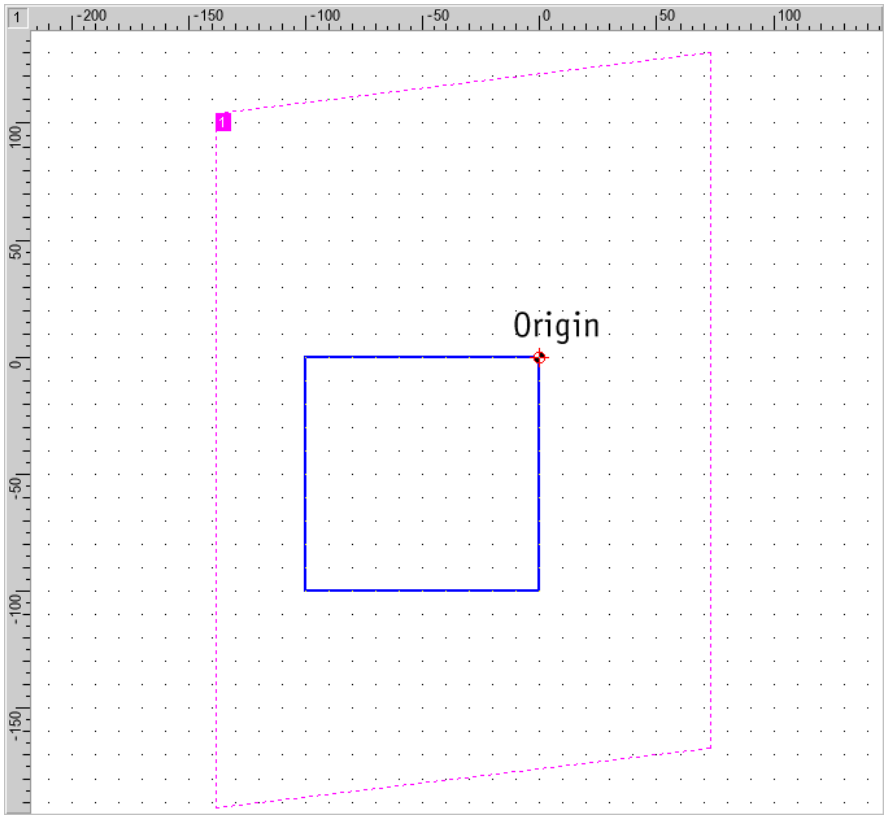


Figure 3

Put the dimetric left view back into window 2 and put the dimetric right view into the main window (Figure 4).

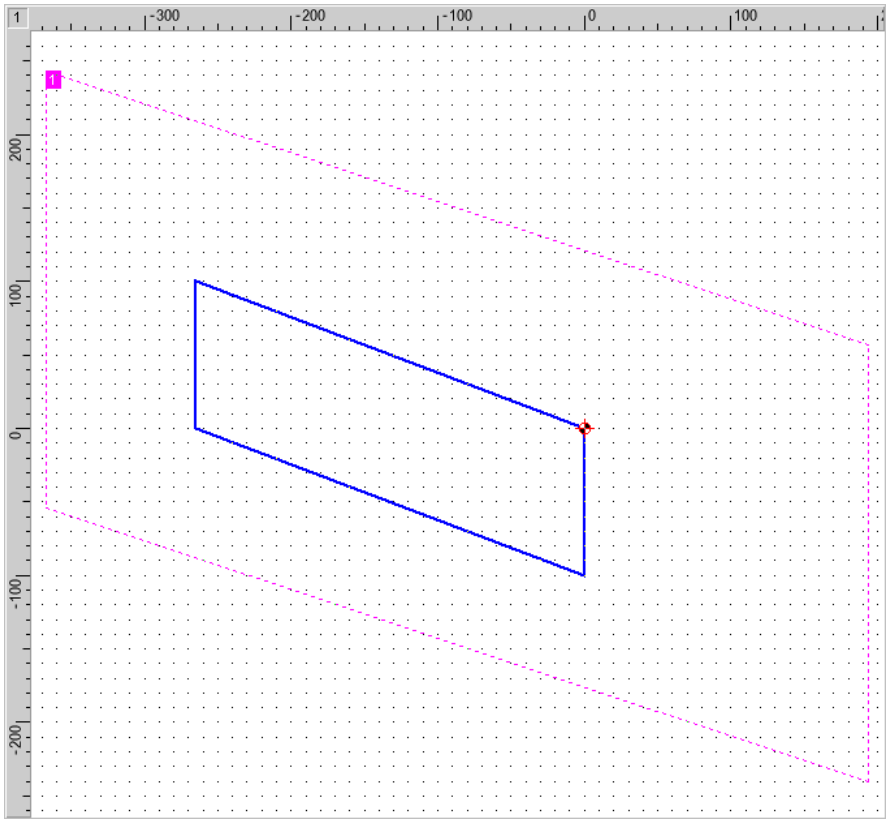


Figure 4

This is the right side of the cube. One of the horizontal sides has already been drawn. Draw another square using **Draw > Polygon > Parallelogram**, placing the first two points on the previously-drawn vertical line and dragging the square's outline right by 100 mm (10 grid points) (Figure 5).

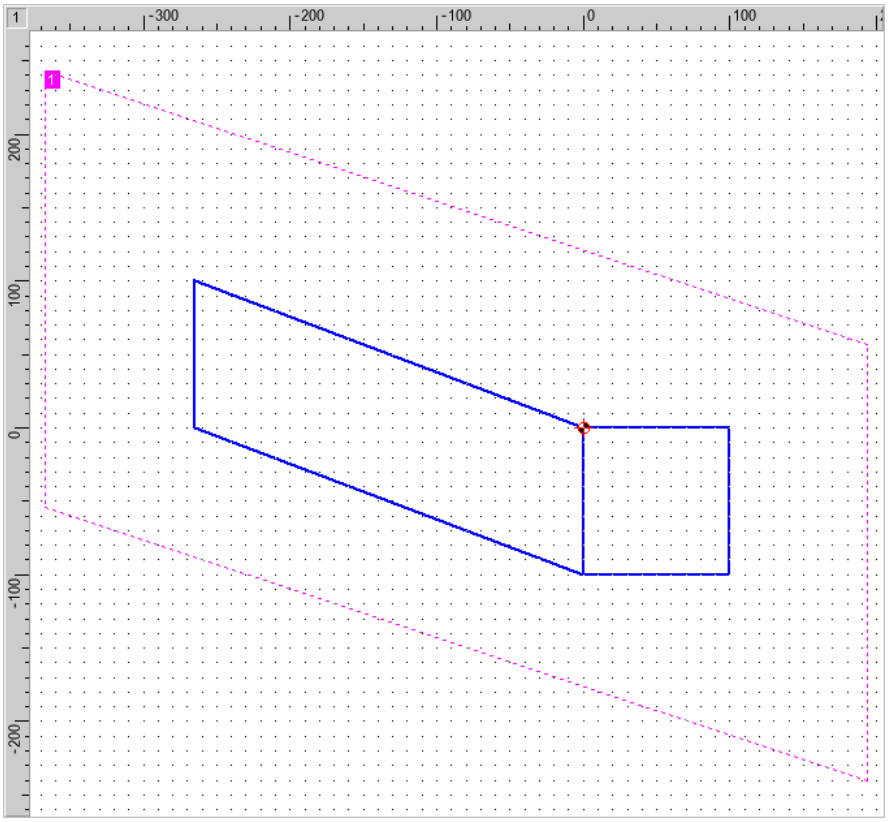


Figure 5

Keep an eye on what is going on in the window with the standard view. A parallelogram is drawn here showing one of the cube's other surfaces.

Put the right view back into window 3 and put the plan view from window 4 into the main window. Again, draw a square on the grid with sides of 100 mm length on the grid. Once again, begin at the marked zero-point and use the vertical line as a guide. You can now see a complete cube in the standard view (Figure 6).

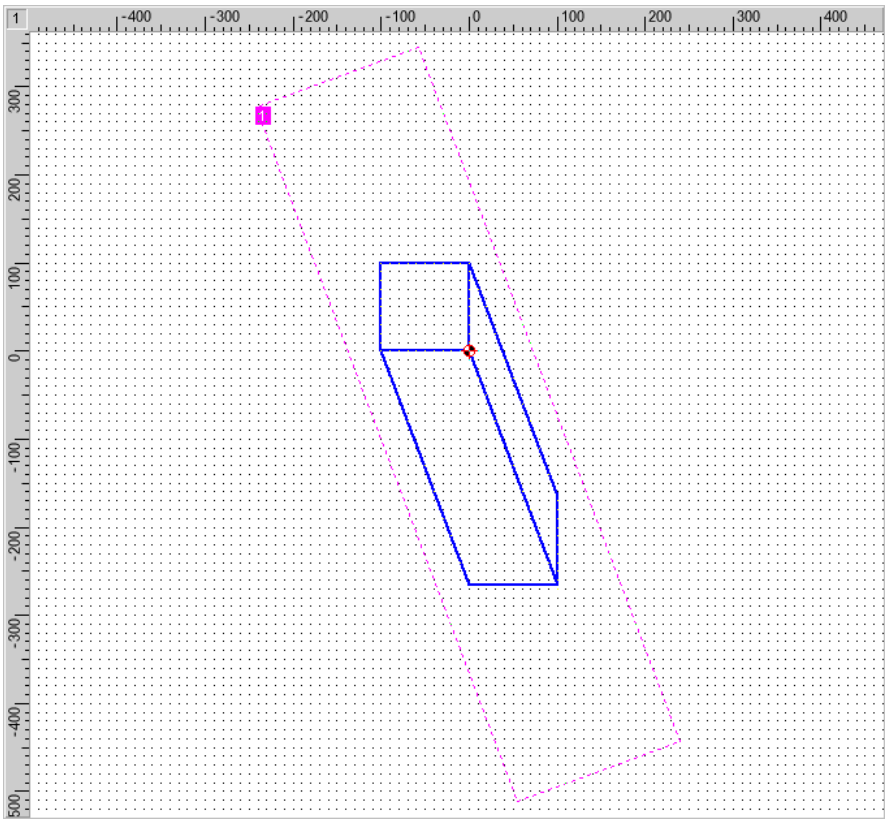


Figure 6

You can use the plan view in the main window for the next step. Choose **Draw > Ellipse > Distorted**. Click at the mid-point of the last-drawn square to specify the center of the ellipse (1). Set the other two points by clicking on the mid-point of the upper horizontal line (2) and the right hand vertical line (3) respectively. In the plan view, a circle will now have been drawn (Figure 7).

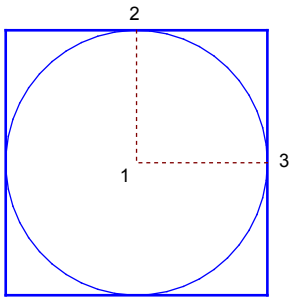


Figure 7

Choose the pen “0.25 mm\\Dash-Dot Line Narrow” and use **Draw > Line > Standard** to draw horizontal and vertical symmetry lines extending 5 mm beyond the square’s edges (that is, one position grid point or half a display point from the edge). Join the corners with diagonals.

Your page should now look like figure 8:

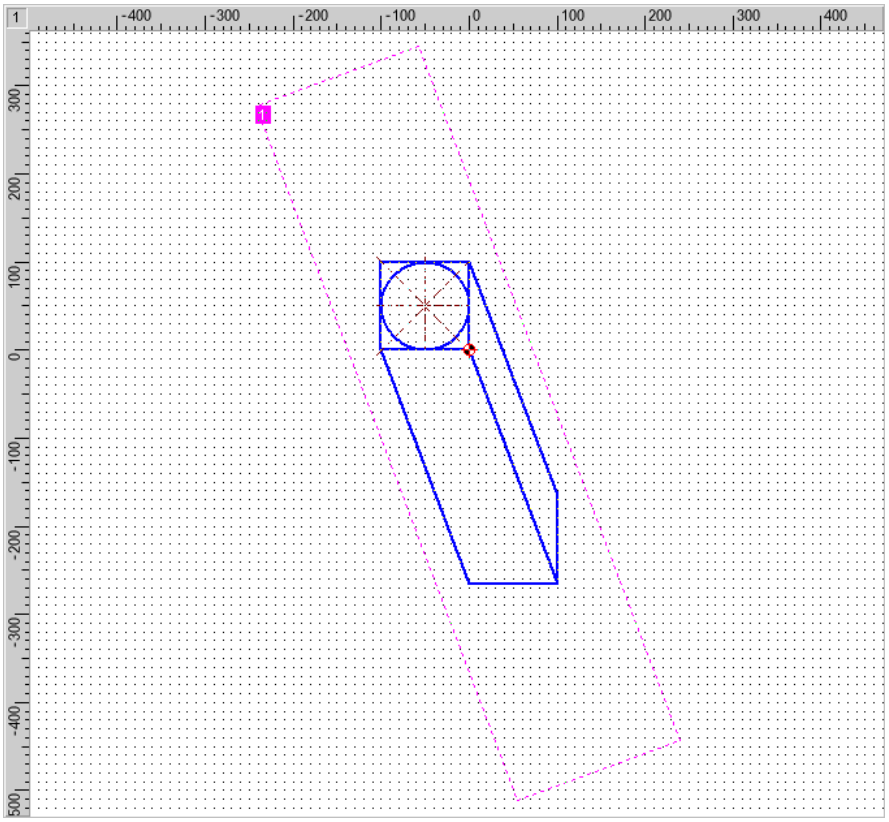


Figure 8

You can now change the view displayed in the main window again and use the same procedure to draw the circle and symmetry lines in the other views.

The finished drawing will now look (depending where you started) something like this:

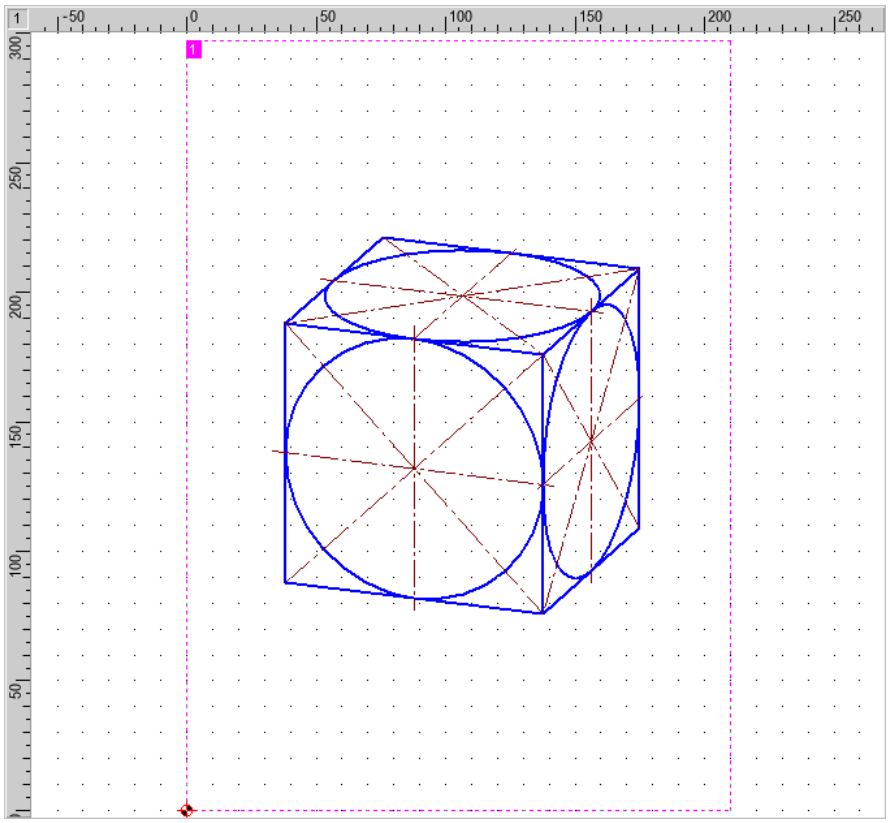


Figure 9

Finally, center the object using **Shape > Align Objects > Centered, Page Both**) and the F10 key, then save it using **File > Save Drawing as** (Figure 10).

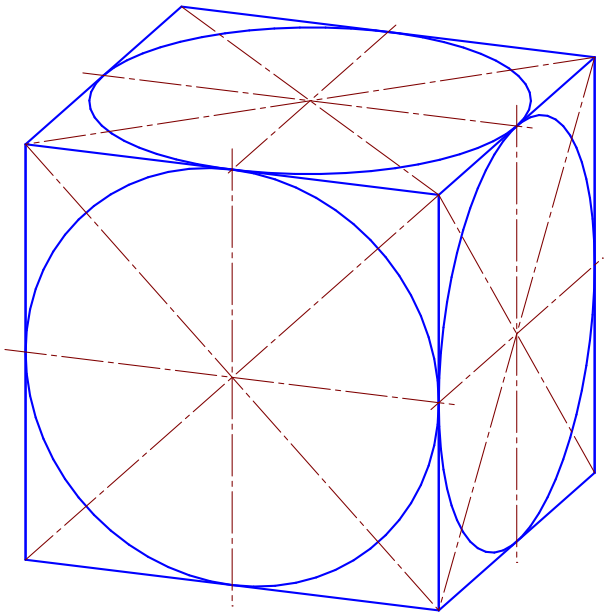


Figure 10

The creation of a drawing in isometric representation or in type 2 dimetric representation (view from the right side) works similar. The only difference is in the views that are chosen for the windows' coordinate systems.

If your screen resolution is high enough, you should use the advantages of multiple window display for all types of drawings. Each window can display another detail of the drawing, allowing fast copy-and-paste operations. For activation of any desired window, simply press SHIFT+(Window Number) on your keyboard.

In case you want to save the multiple window environment for isometric representation or other purposes, create a special template file using **View > Template > Save as**.